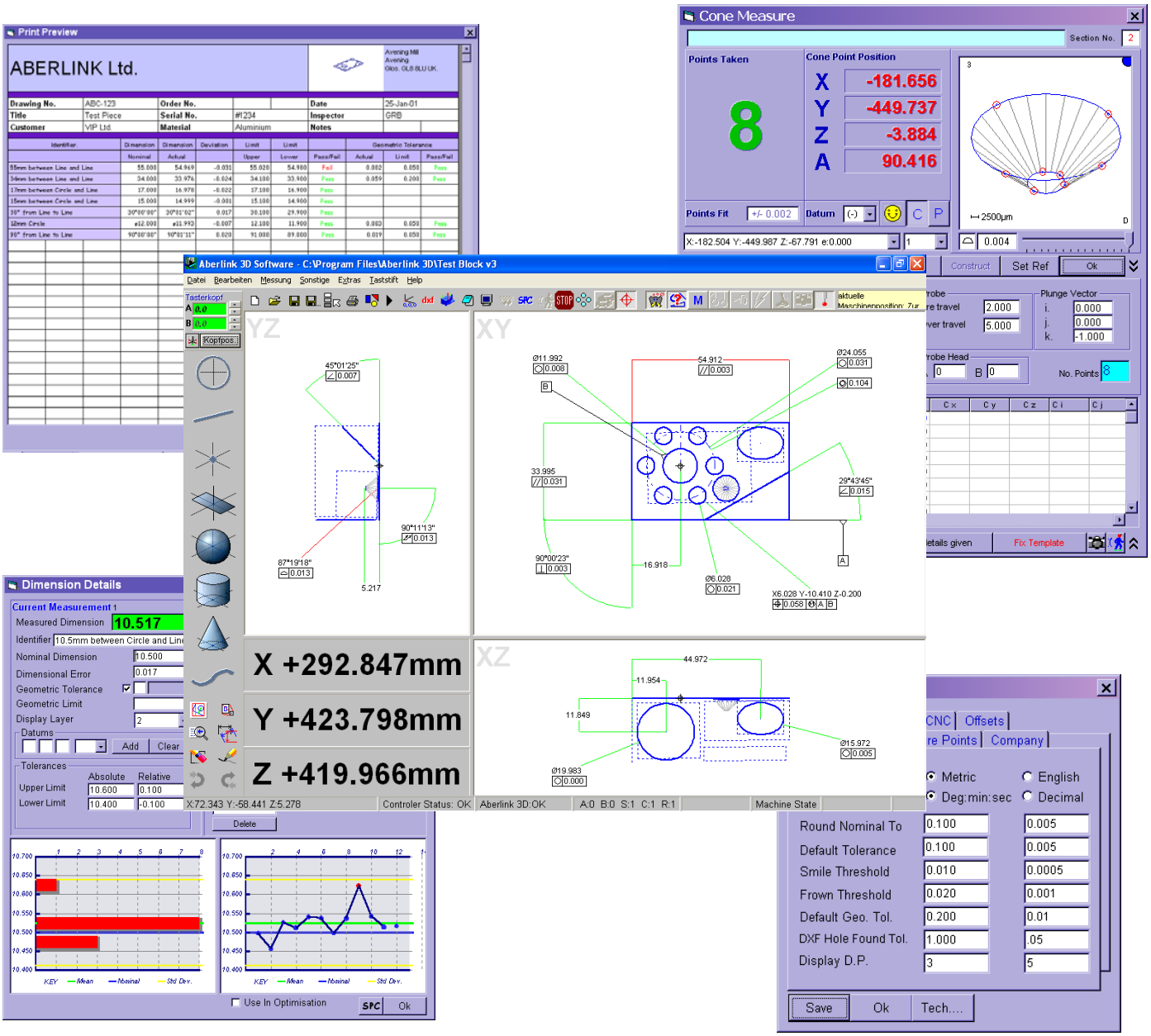
ABERLINK 3D MEASUREMENT SOFTWARE MANUAL

(FOR FARO ARMS)



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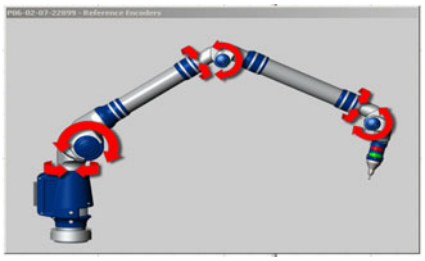
1.0 STARTING ABERLINK 3D

Upon starting the Aberlink 3D software a Faro Arm (USB) window will also be opened by default. This window cannot be closed whilst running the Aberlink 3D software, and on the close command, or by clicking on the Aberlink 3D software in the background this window will be minimised. The functionality available within this window is discussed below.

Please note that when using the Faro Gage, the Gage software must already be running on the PC before opening the Aberlink 3D software.

Aberlink 3D will automatically try to 'connect' to the Faro arm upon start up.

If the Faro arm has not been referenced, the following window may appear:



Each of the angular encoders within the arm must be moved through a reference point so that the arm knows where it is. By moving each joint on the arm each of the red arrows will start to disappear. When all of the encoders are referenced this window will disappear.

If using the Faro Gage, this operation may have to be performed within the Gage software.

1.1 Summary of Arm button usage

The Faro arm has two pairs of buttons. The 'Front' buttons are coloured green. The 'Back' buttons are coloured red.

Aberlink 3D considers the arm to be in one of two modes at any time:

- Measure Mode – the arm is being used to take measurements
- Mouse Mode – the arm is being used as a mouse

Mouse mode can only be used if the ‘Enable mouse’ checkbox is ticked within the Faro Arm (USB) window (see below).
In **Measure Mode**, buttons are used as follows:

- Front (green) Button on its own is used to take a point.
- Front (green) Button pressed while the Back (red) button is held down is used to retake a point.
- Back (red) Button pressed twice in quick succession switches to Mouse Mode
- Back (red) button held down for a long time is used to OK the current feature.

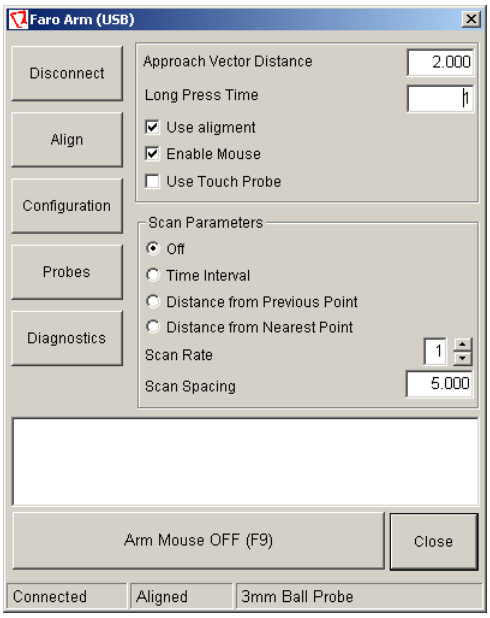
In **Mouse Mode**, buttons are used as follows:

- Front (green) Button on its own acts as the left mouse button
- Back (red) Button pressed twice in quick succession switches to Measure Mode

In addition, the back button performs the action configured in the Faro configuration form. For use with Aberlink 3D we recommend that the ‘Back Button’ is configured as ‘Centers Cursor’ or ‘Application Specific’.

1.2 The Faro Arm (USB) Window

The ‘Faro Arm (USB)’ form is used to configure operation of the arm with Aberlink 3D software. If the form is not open it will be minimized on the Windows task bar. Click the icon on the task bar to make the window visible:



1.2.1 Disconnect / Connect

This button can be used to disconnect the Faro Arm. This will not normally be required. When disconnected, this button changes to a ‘Connect’ button.

1.2.2 Align

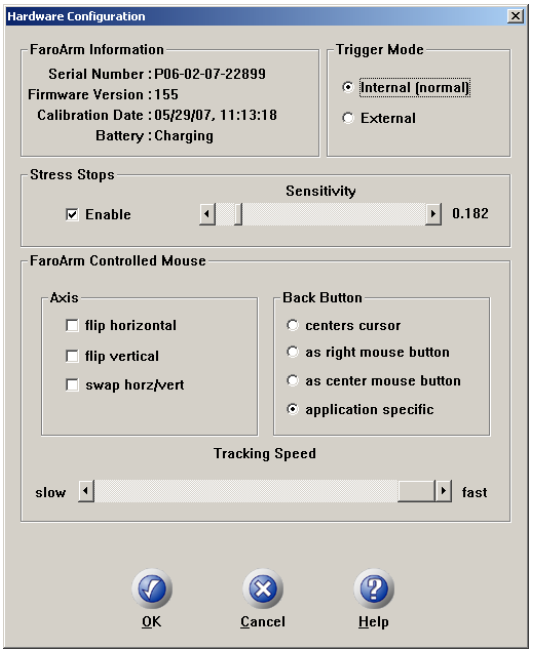
This button brings up the Alignment form that allows the user to define a new (approximate) reference frame by taking measurements at the new origin, at a point on the positive X axis and at a point on the XY plane with a positive Y

value. Once an alignment has been set up, the ‘Use alignment’ check box is enabled and ticked. Whilst the check box is ticked, all measurements taken will be mapped from raw arm coordinates to the new reference frame. The user can set up a new alignment at any time (using the Align button) or disable an existing alignment by un-checking the ‘Use alignment’ check box.

Note that this function is generally used to define an approximate alignment frame – with final alignments set by measuring features and setting them as references within the Aberlink 3D measurement software in the usual way.

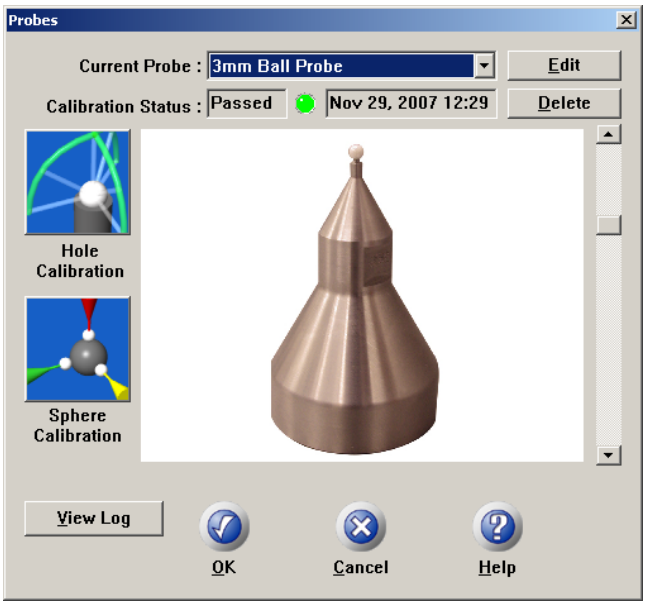
1.2.3 Configuration

Pressing this button brings up the Faro configuration form. See Faro documentation for the use of this form. For use with Aberlink 3D we recommend that the ‘Back Button’ is configured as ‘Centers Cursor’ or ‘Application Specific’.



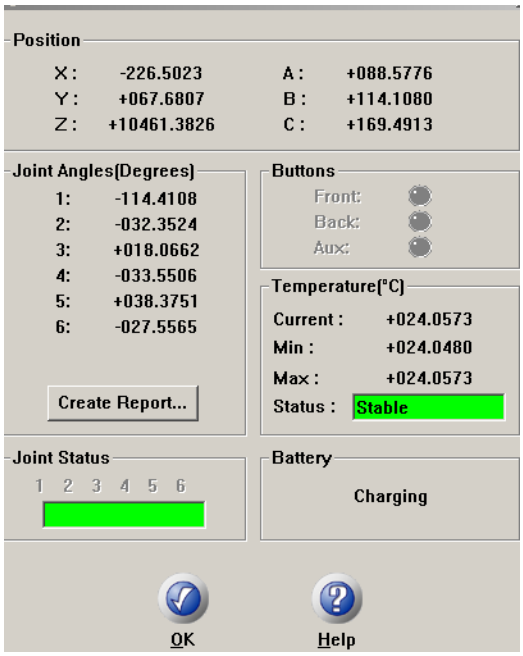
1.2.4 Probes

Pressing this button brings up the Faro probes form. See Faro documentation for the use of this form. Use this form to select the current probe and to initiate probe calibration. When you close this form, Aberlink 3D will automatically recognise which probe is in use and use the correct stylus parameters.



1.2.5 Diagnostics

Pressing this button brings up the Faro diagnostics form. See Faro documentation for the use of this form.



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1.2.6 Enable Mouse

This checkbox can be used to enable/disable the arm's remote mouse functionality. If the checkbox is ticked, remote mouse functionality is enabled.

1.2.7 Approach Vector Distance

For correct probe tip diameter compensation, Aberlink 3D uses the 'approach vector' to the measurement, i.e. the direction in which the probe moved to reach the measurement point. The approach vector is calculated from the measurement point and a recent probe position that is at least the 'approach distance' away from the measured point. When a measurement is taken with the arm, it is common for the probe to 'skate' a small distance on the surface around the measurement point. Since small probe movements less than approach distance do not affect the approach vector, these skating movements do not matter. A typical value for this setting is 2mm.

1.2.8 Long Press Time

In measuring mode, a 'long' press of the 'Back' button has the same effect as clicking the OK button on the 'Measure' form (i.e. to close the current feature). It is not recommended to set this value to less than 1 second since the 'Back' button is also used to retake bad points. Pressing and holding the 'Back' button, then briefly pressing the 'Front' button before releasing the 'Back' button will initiate 'Retake'. If the 'Long Press Time' is set too low, it will be difficult to indicate the 'Retake' before the 'OK' action is recognised.

1.2.9 Use Touch Probe

Tick this box if the arm is being used with a Renishaw touch trigger probe. Scan mode is disabled if this box is ticked and also the Front, green button cannot be used to take points (but it can still be used in conjunction with the back, red button to retake points).

1.2.10 Scan Parameters

In this context, 'scanning' means taking multiple measurements by holding down the 'Front' button and moving the probe.

Off

To disable scanning, choose this option.

Time Interval

In this mode, Aberlink 3D will take a measurement at a regular intervals whilst the 'Front' button is depressed, at a rate controlled by the 'Scan Rate' setting. This may be set from 1 (one measurement per second) to 10 (ten measurements per second).

Distance from Previous Point

In this mode, Aberlink 3D will take a measurement when the probe has moved by at least the 'Scan Spacing' from the previous point and whilst the front, green button is depressed. The spacing is set in mm or inches, depending on the whether Aberlink 3D is configured for metric or imperial units.

Distance from Nearest Point

In this mode, Aberlink 3D will take a measurement when the probes is at least the 'Scan Spacing' from the nearest point already measured in the current feature whilst the front, green button is depressed. Again the spacing is set in mm or inches, depending on the whether Aberlink 3D is configured for metric or imperial units.

1.2.11 Mouse Control Button

This button can be used to change between Measure mode and Mouse mode. Its caption changes to indicate what will happen if it is pressed. The same effect can be obtained by pressing the F9 key on the keyboard.

1.2.12 Close

This button causes the Faro form to be minimised on the task bar.

1.2.13 Event History

The window to the left of the Close button displays a history of recent arm events. For measurements, the coordinates are shown and also the approach vector. The most recent event is at the top of the window.

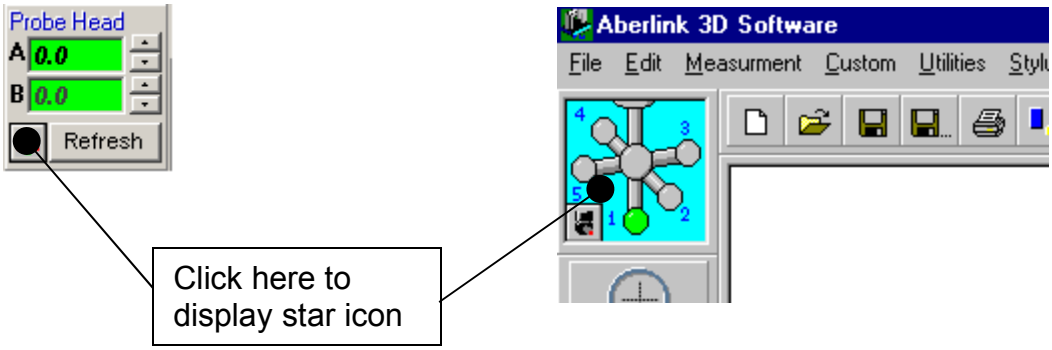
1.2.14 Status Bar

The status bar at the bottom the Faro Arm form shows the connection status, the alignment status, and the probe details.

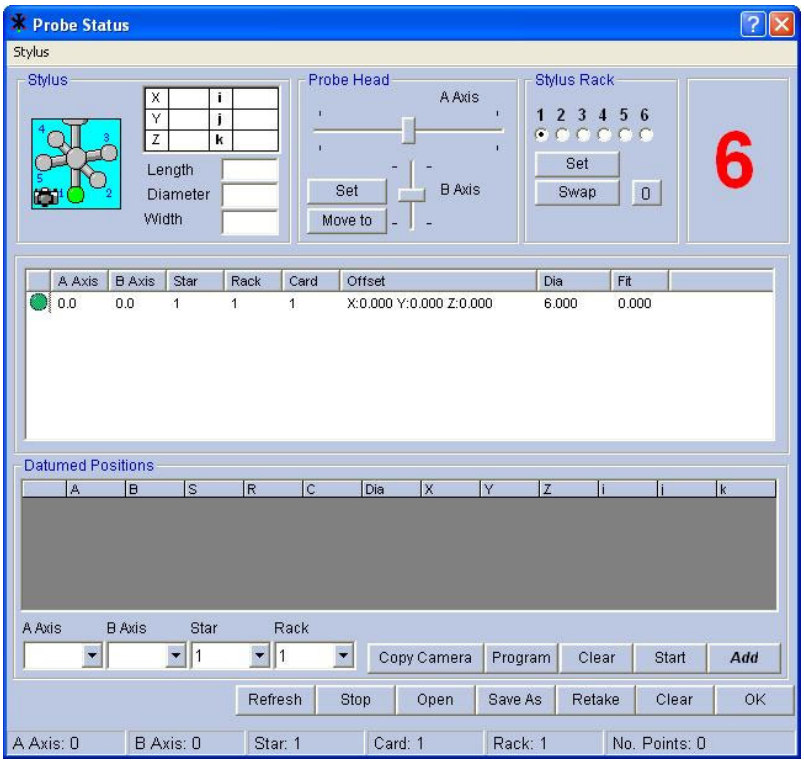
1.3 Setting the Stylus Diameter within Aberlink 3D

In the Probe Head button (top, left corner of the Main Aberlink 3D screen), the probe head position can be displayed either as an A and B position (for indexible heads only and therefore not used with arms) or as a star stylus. With an arm it is recommended that the star stylus is displayed, with only position 1 (straight down) used. This position should be shown in green.

If the software is showing an A and B position, click on the small star stylus icon, to change the view, as follows:



To bring up the Probe status Window click on the light blue background area behind the star icon:


Probe Status
Stylus
Probe Head
A Axis
B Axis
Set
Move to
Stylus Rack
1 2 3 4 5 6
Set
Swap
6
Length
Diameter
Width
A Axis B Axis Star Rack Offset Dia Fit
0.0 0.0 1 1 1 X:0.000 Y:0.000 Z:0.000 6.000 0.000
Datumed Positions
A B S R C Dia X Y Z i j k
A Axis B Axis Star Rack
Copy Camera Program Clear Start Add
Refresh Stop Open Save As Retake Clear OK
A Axis: 0 B Axis: 0 Star: 1 Card: 1 Rack: 1 No. Points: 0

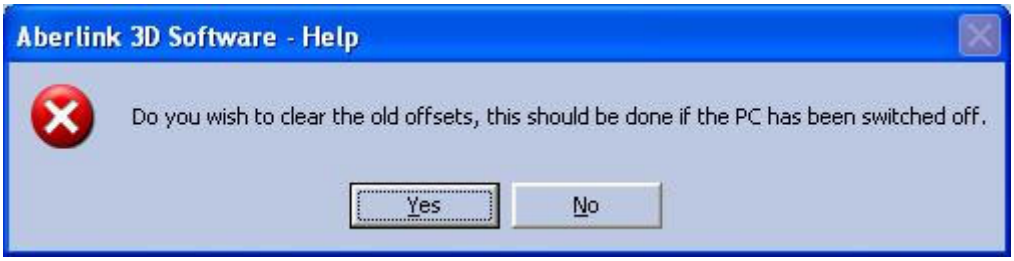
This is how the Probe Status window should look when using a 6mm probe

When using a Faro arm there should only be one current stylus offset, as shown above. The X, Y and Z values should all be zero and the diameter should relate to the diameter of the stylus being used.

When setting up the software for the first time, if there is more than one stylus offset present, the current stylus offset will be shown with a green dot, and the other stylus offsets will have a red dot. Right click on the red dots and choose 'clear data' from the menu to remove the additional offsets.

If the wrong diameter is initially selected this can be edited as follows: Right click on the green dot adjacent to the current offset and select 'edit' from the menu. This will bring up the following window:

Simply type in the correct diameter and click OK. This will bring up the following window:



Choose 'Yes'

> *Note – this option can also be used for clearing out any unwanted stylus offsets.*

Then OK the Probe Status window.

> *Also Note – that when you change the probe size from within the Faro Arm (USB) window using the 'Probes' option, the diameter will automatically be updated within the Aberlink 3D Probe Status window.*

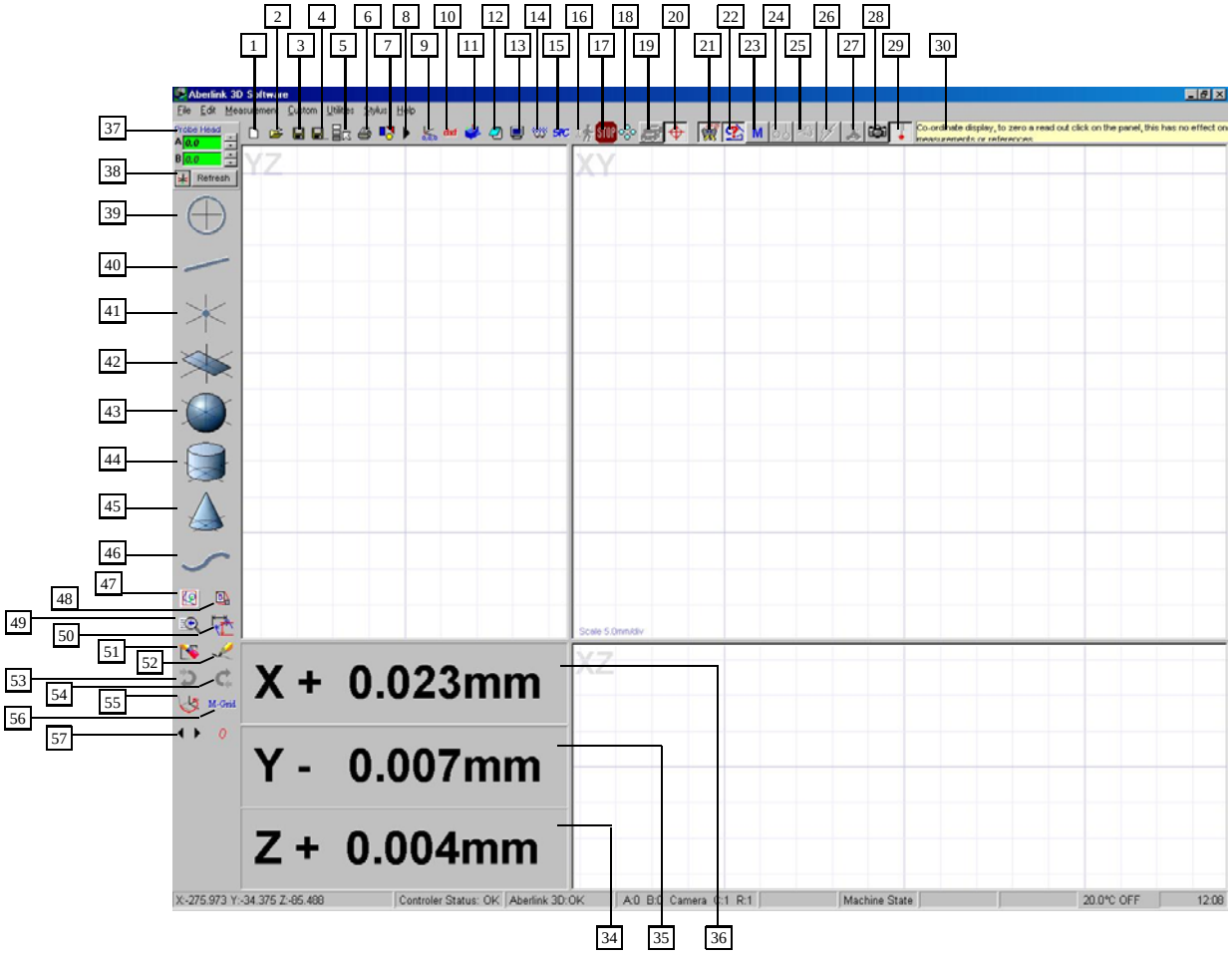
2.0 ABERLINK 3D SOFTWARE

2.1 Exploring The Main Screen

The Aberlink 3D software has been written to be extremely intuitive based around a graphical interface, operating under a Windows operating platform.

As a component is inspected, a graphical representation of it is built up in the three 2 dimensional and an isometric view in the main screen. All the functionality of the software is available from the main screen, and this will only ever bring up one further layer, or window. In this way the screen can never become too confusing, as closing the top window will always return to the main screen.

It is therefore worthwhile taking a few minutes to become familiar with the buttons available in the main screen.



1. New Inspection
   Click on this button when you wish to start a new inspection.

2. Open File
   Click on this button when you wish to recall a previously saved inspection.

3. Save File
   This button will save an inspection under its existing name.

4. Save As
   Use this button when you wish to save an inspection giving it a new name.

5. Name & save File
   Use this button when you wish to save an inspection with password protection.

6. Print
   Click on this button when you wish to produce a hard copy inspection report.

7. Set Up
   For setting the 'user definable' parameters within the software.

8. Play
   Prompts the measurement sequence (program) for multiple components.

9. Reference Machine
   Not relevant for a Faro arm.

10. DXF File & Curve File save / open
    Allows full import and export functionality with DXF files.

11. Import CAD model
    This button allows you to import a solid CAD model and measure against it (this is an optional software module)

12. Inspection Notes
    This allows text and images to be stored with inspections.

13. Display Drawing Layers
    This button turn layers on and off the 'Display Drawing Layers'

14. Workpiece Co-ordinate
    Not relevant for a Faro arm.

15. External SPC Data Link
    This button allows you export data to third party SPC software if the link is available.

16. Jog Machine
    Not relevant for a Faro arm.

17. STOP button
    Can be used to stop a program before the end.

18. Ring Array
    Not relevant for a Faro arm.

19. Park CMM
    Not relevant for a Faro arm.

20. Geometric Tolerance
    This button will display relevant geometric tolerances on the screen.

21. Store Points
    This function will store measurement points and then apply them to a feature when called up retrospectively

22. Feature Predict
    This function will predict what feature to fit to taken measurement points.

23. Display measured Details or template Details
    This button switches the display between template and measured details

24. Display probe points
    Not relevant for a Faro arm.

25. Switch on / off Air
    Not relevant for a Faro arm.

26. Switch on / off Motors
    Not relevant for a Faro arm.

27. Switch on / off Joystick
    Not relevant for a Faro arm.

28. Switch on / off Camera
    Not relevant for a Faro arm.

29. Switch on / off Touch Probe
    This button will turn on and off a touch trigger probe if one is fitted to the arm.

30. Tool Tip Bar
    Gives useful hints when performing functions.

31. XY Graphics Area
    The XY view of the measured features will appear in this area.

32. XZ Graphics Area
    The XZ view of the measured features will appear in this area.

33. YZ Graphics Area
    The YZ view of the measured features will appear in this area.

34. Z Axis Readout
    Can display machine co-ordinates, component co-ordinates or incremental position, and can be used for performing quick check measurements.

35. Y Axis Readout
    Can display machine co-ordinates, component co-ordinates or incremental position, and can be used for performing quick check measurements.

36. X Axis Readout
    Can display machine co-ordinates, component co-ordinates or incremental position, and can be used for performing quick check measurements.

37. Probe Datum Button
    This is used to datum new styli and probe positions.

38. Probe Head Icon
    This will change the Probe Datum Button between probe head A & B angles and the star stylus .

39. Circle Measure Button
    Used when measuring or constructing arcs, holes or circular bosses.

40. Line Measure Button
    Used when measuring or constructing lines.

41. Point Measure Button
    Used when measuring or constructing single points.

42. Plane Measure Button
    Used when measuring or constructing planes (flat faces).

43. Sphere Measure Button
    Used when measuring internal or external spheres.

44. Cylinder Measure Button
    Used when measuring holes or circular bosses as a cylinder (in 3 dimensions).

45. Cone Measure Button
    Used when measuring conical shapes (in 3 dimensions).

46. Curve measure Button
    Used when measuring curves & profiles

47. Zoom Full
    Will zoom the graphics areas to cover the full size of the table.

48. Zoom Out
    Will zoom the graphics areas out by a fixed percentage.

49. Zoom Last
    Will return the graphics areas to their last view.

50. Dimension Type
    Can be used to define the type of dimension brought up onto the screen.

51. Erase
    Can be used to remove any given feature or dimension.

52. Auto Zoom Redraw
    Will automatically zoom the graphics areas to fit the existing features and dimensions.

53. Undo
    Will undo the last action performed.

54. Redo
    Will redo the last action performed.

55. Grid Shift
    Allows a rotation or translation of the measured features, or an optimum alignment to minimise the errors for selected dimensions.

56. Machine Grid
    Cancels all references that have been set, and returns to the machines own reference position.

57. Feature Select
    Allows specific features to be selected, even when overlapping.

3.0 THE PLANE UNIT

The plane unit is used to measure flat surfaces, such as one face of a cube, by taking 3 or more measurement points spread across the surface.

Planes can also be constructed through 3 or more previously measured points.

You can also construct a plane, offset from another plane by a fixed distance, which can be entered via the keyboard.

Also a mid-plane between 2 previously measured planes can be constructed.

3.1 Measuring Planes Directly

Background

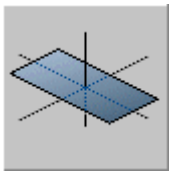
Planes are used to measure flat surfaces on components, and the software will construct the 'best fit' plane through the measurement points taken.

In order to apply the stylus compensation in the correct direction, the software will look at the direction of motion of the probe when taking the measurement points.

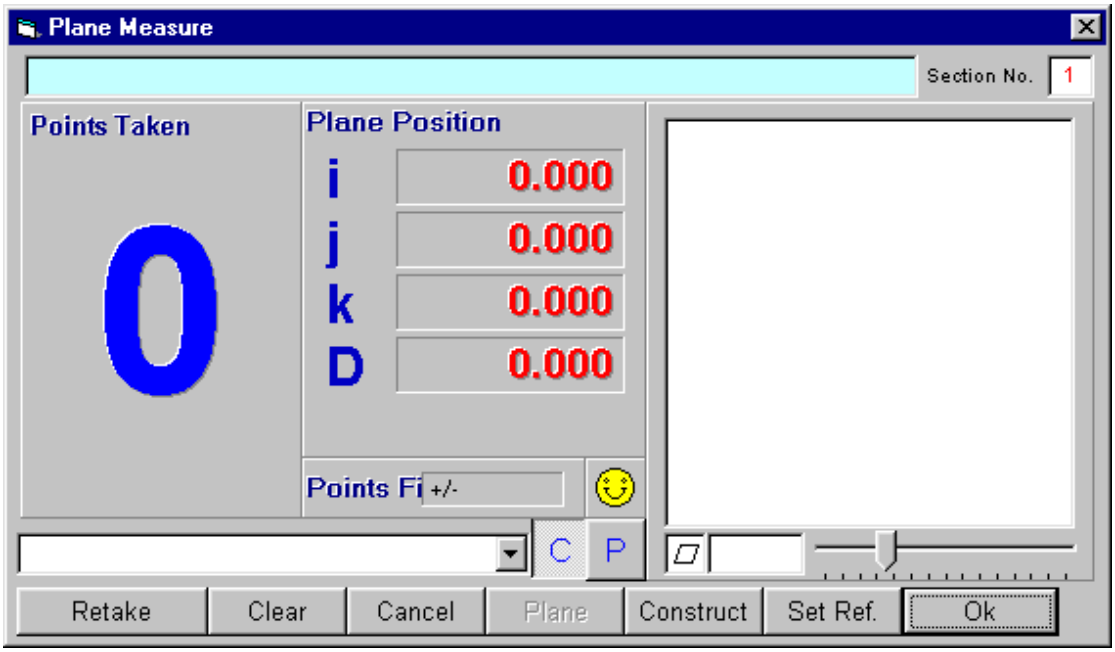
If you measure a plane and set it as a reference, this will align the component to this plane, ie. You will have defined 3 of the 6 degrees of freedom for the component (see chapter 5 – Aligning the axes of a component). .

Method

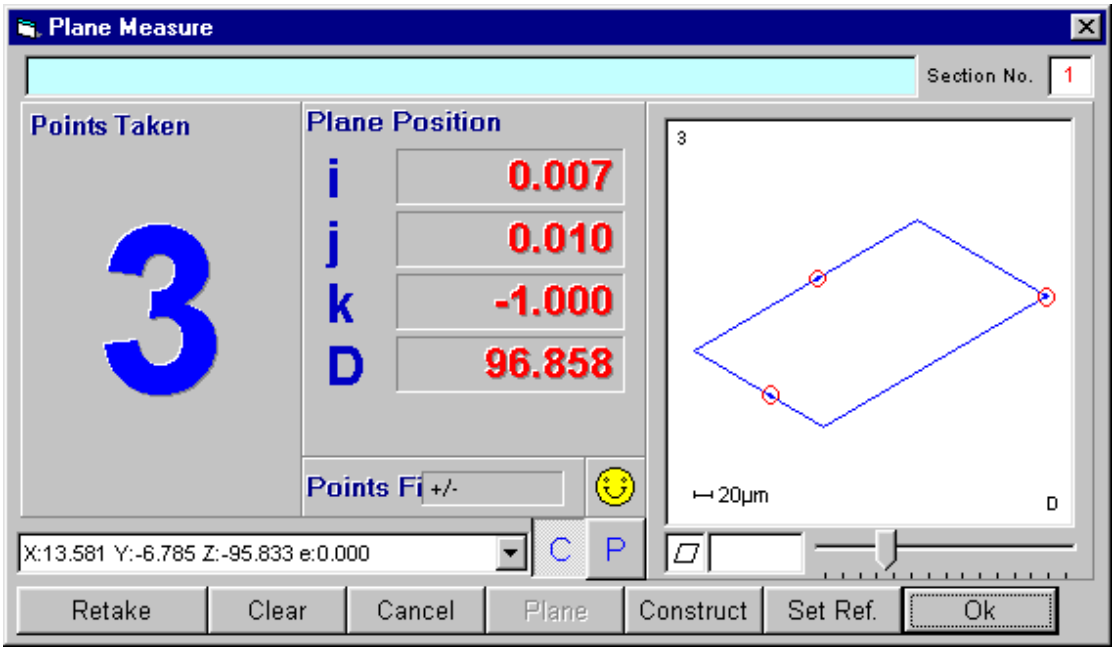
To measure a plane, click on the Plane Measure button, from the main screen:



This will bring up the Plane Measure window:



Now take measurement points on the surface that you wish to measure. After 3 points a pictorial representation of the plane will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The I,J and K values displayed represent the direction vector of the normal to the plane and the D value represents the distance between the plane and the current origin.

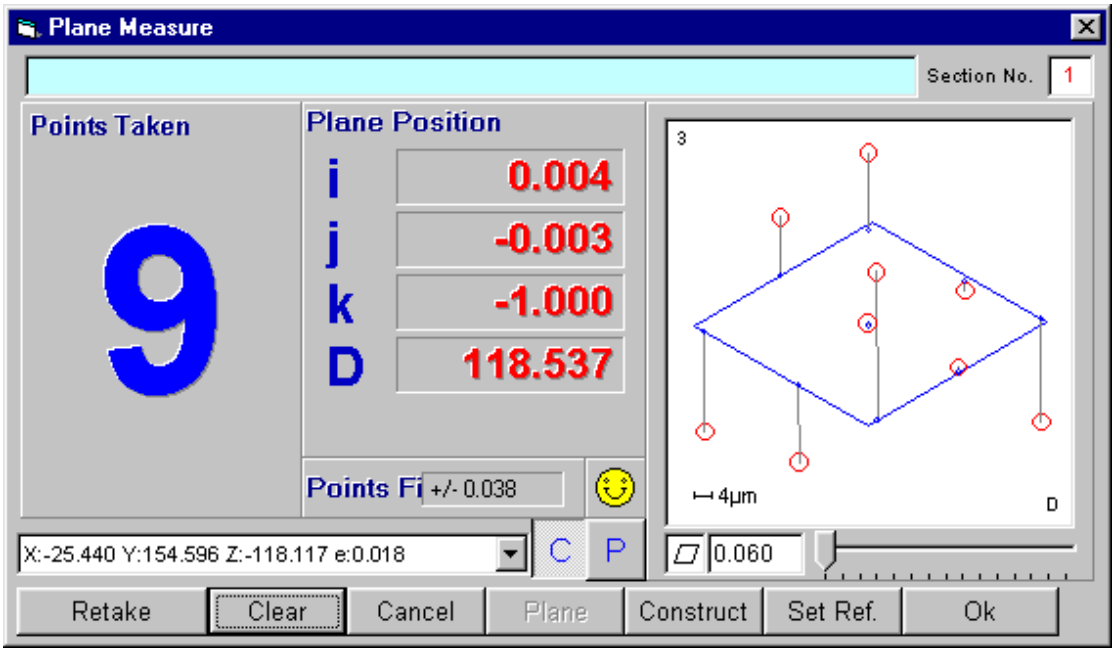


A plane can always be calculated to perfectly fit through three points, therefore after the first 3 points the flatness value for the plane, shown under the graphical representation, will be blank.

It is therefore good practice to take at least one more point. On the fourth point the software will now be able to calculate a value for the fit of the points (3 sigma) on its calculated 'best fit' plane and a value for the geometric flatness of this plane.

In general the more points taken on a plane, the more information can be gleaned, and a minimum of 4 points is recommended. Similarly the points taken should cover as much of the surface of the plane as possible.

The graphical representation of the plane shows the points taken as a red circle with a blue isometric rectangle representing the 'best fit' plane through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.



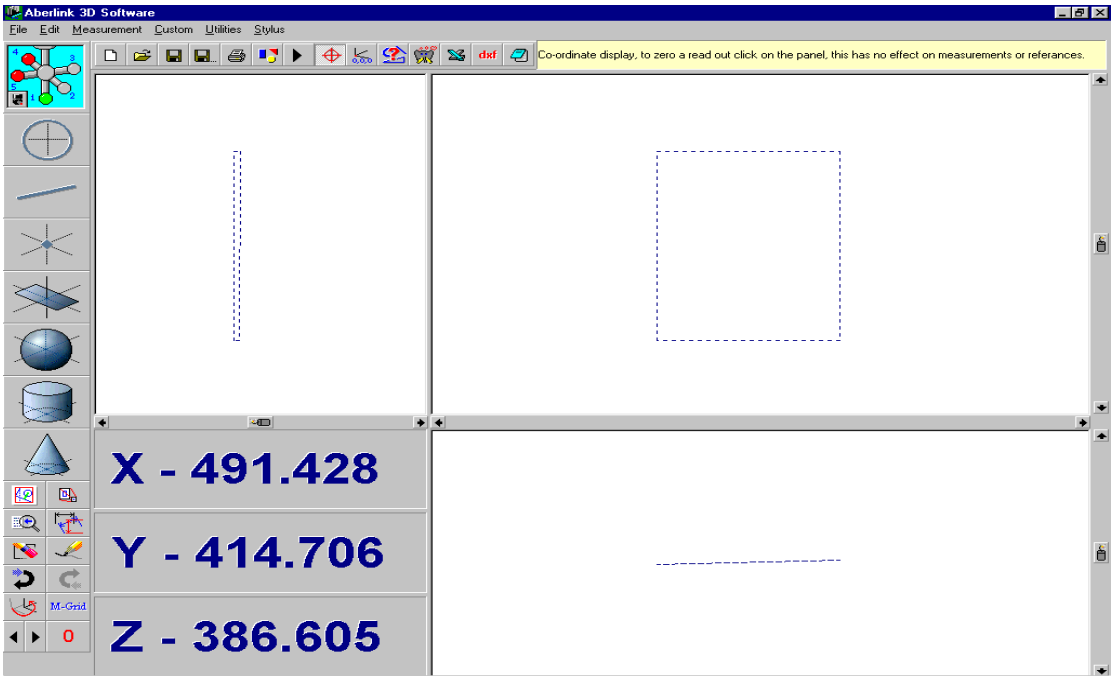
If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points - see chapter 13 - Erasing Data Points.

3.1.1 When to OK a plane, and when to reference it.

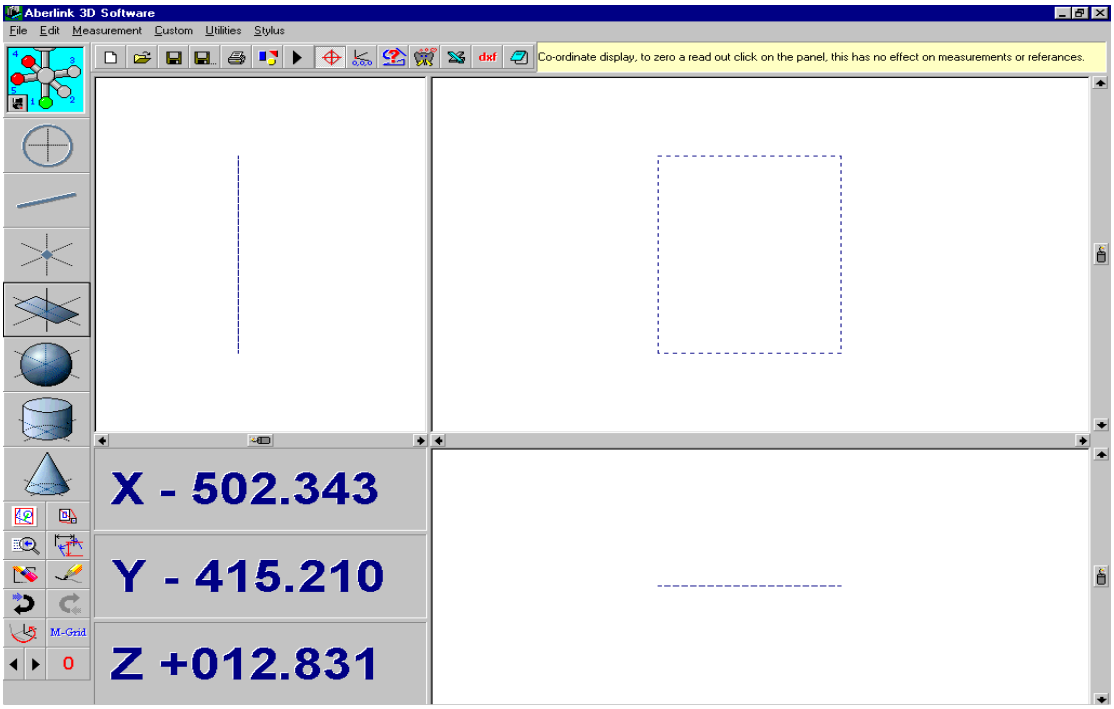
Having measured a plane, you can either click on the 'OK' button or the 'Set Ref ' button followed by OK.

When you OK the plane, the measurement window will close, and the program will return to the main screen.

The graphical representation of the plane should be shown as a dashed rectangle in all of the 3 2-dimensional views. This representation will be pictured at whatever angle the plane is positioned at relative to the axes which have already been set up.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 491.428
Y - 414.706
Z - 386.605
M-Grid
0

If you set the plane as a reference plane by clicking on ‘Set Ref’ before you click the ‘OK’ button then the graphical representation of the plane will again be shown as a dashed rectangle, but this time it will be aligned to the plane.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 502.343
Y - 415.210
Z +012.831
M-Grid
0

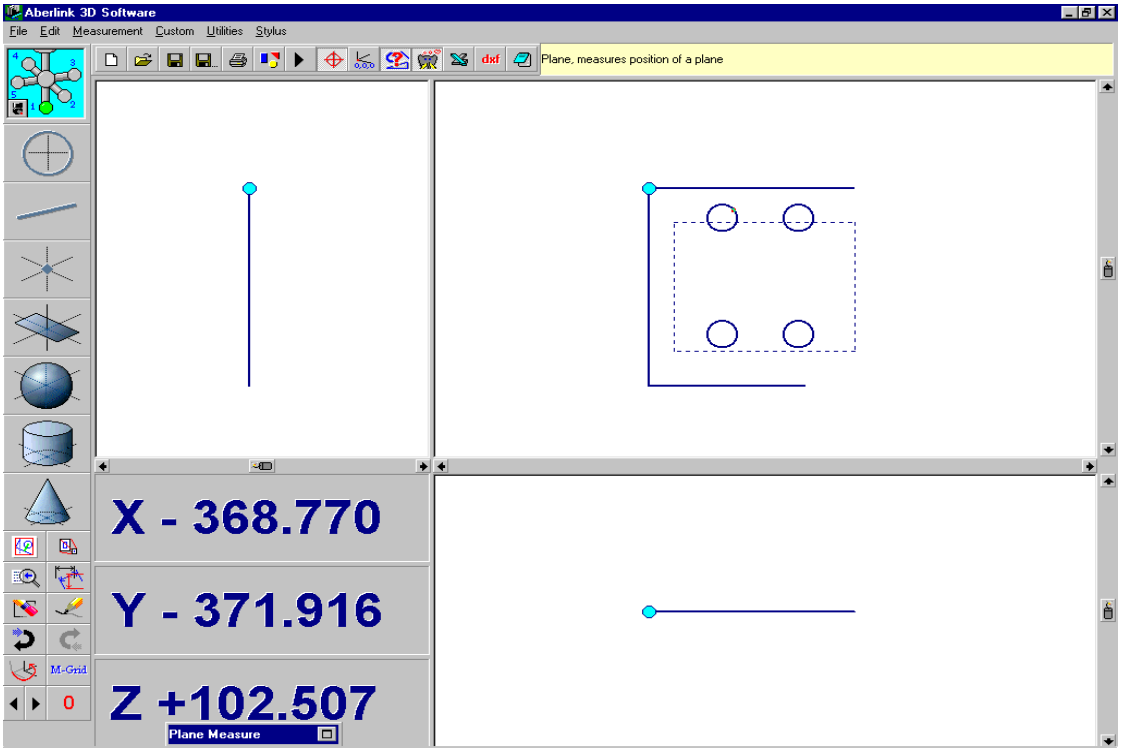
Note *– Only one plane can ever be set as a reference at any point in time. If a second plane is set as a reference, this will override the initial reference plane, and the graphical representation will now re-align itself to the second plane.*

3.2 Constructing Planes

3.2.1 Constructing a Plane through previously measured points

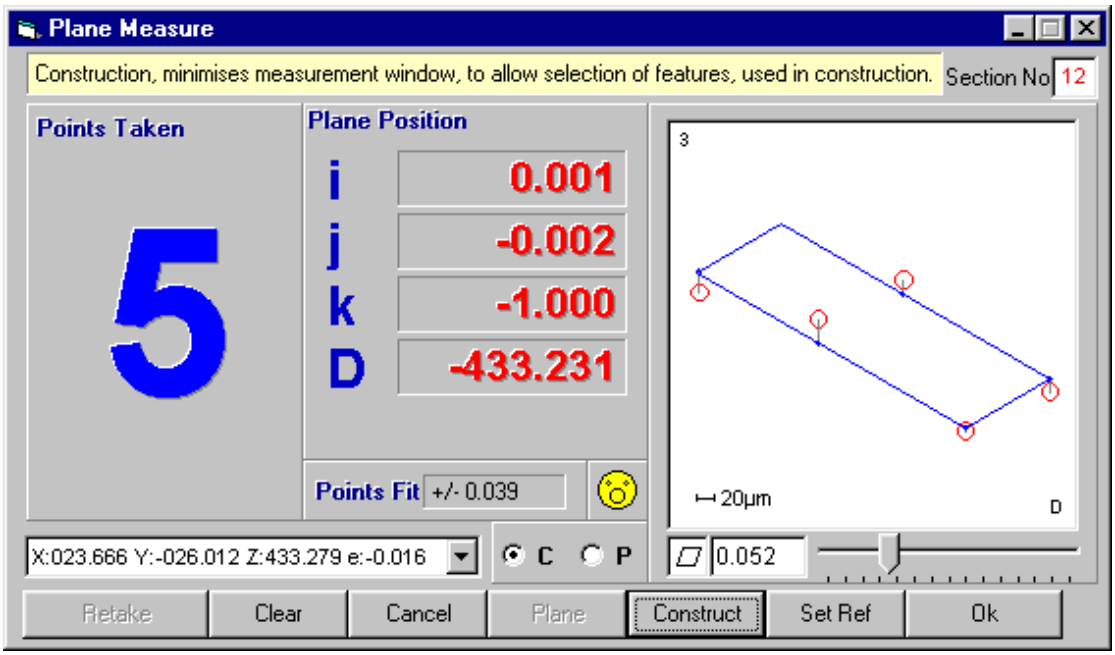
As well as measuring planes directly, they can also be constructed through features which have previously been measured, or constructed. For this purpose, circles and spheres may be considered as points, and lines, cones and cylinders can be considered as 2 points at the ends of their axis. A minimum of three points is required to construct a plane.

To construct a plane, click on the Plane Measure button, from the main screen. This will bring up the Plane Measure window. Now click on the ‘Construct’ button. The Plane measure window will now shrink to a small box at the bottom of the screen:



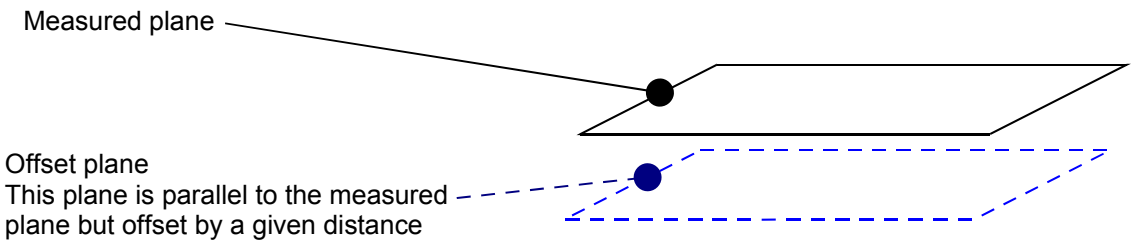
Now simply select the features through which you wish to construct the plane by clicking on them. After you have selected the first feature a prompt will appear, to confirm what is happening:

Click on OK, and then continue to click on the remaining features. When you have selected all the desired features bring the Plane Measure window back by clicking on the right hand end of the shrunken box. A graphical representation of the plane will appear with values for the fit and flatness given if more than 3 points were selected.

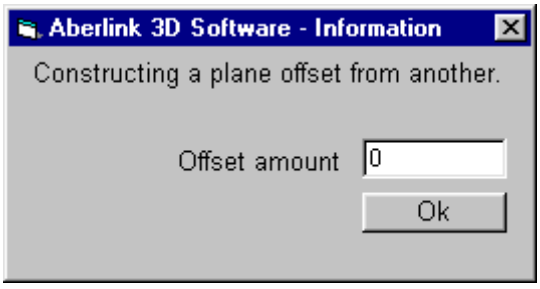


You may now click on ‘OK’ or ‘Set Ref’ as appropriate, as for any plane. The software will now return to the Main Screen.

3.2.2 Constructing a Plane offset from another Plane

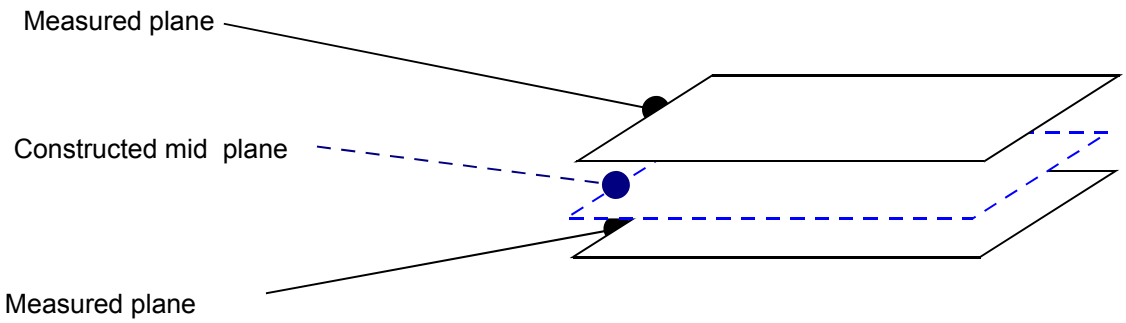


A plane may be constructed which is offset from an existing plane by a fixed distance. To construct such a plane, click on the Plane Measure button, from the main screen. This will bring up the Plane Measure window. Now click on the ‘Construct’ button and the Plane Measure window will minimize. Click on the previously measured plane twice. This will bring up the following window:

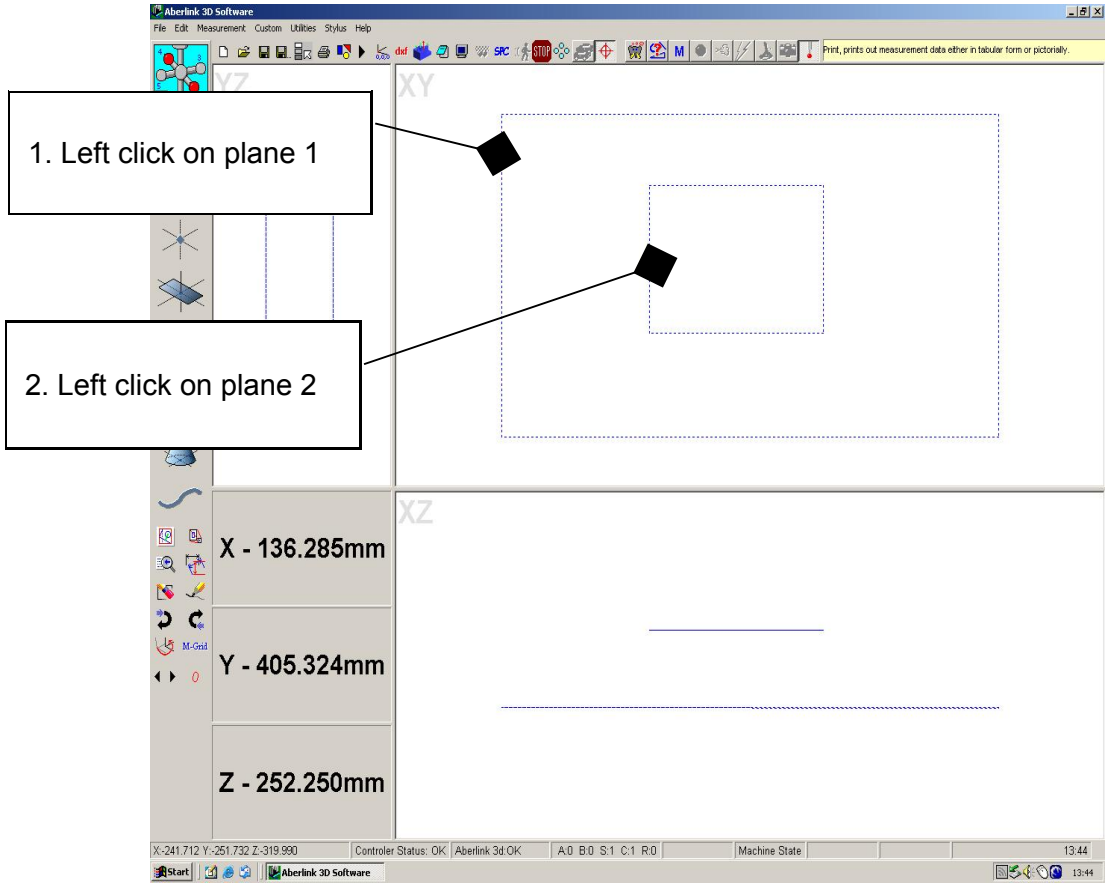


Enter the desired offset, and then click ‘OK’. This will bring back the Plane Measure window. Click ‘OK’ again. The screen will now return to the Main Screen, with the new plane shown offset from the original by the entered amount.

3.2.3 Constructing a mid plane between two measured planes

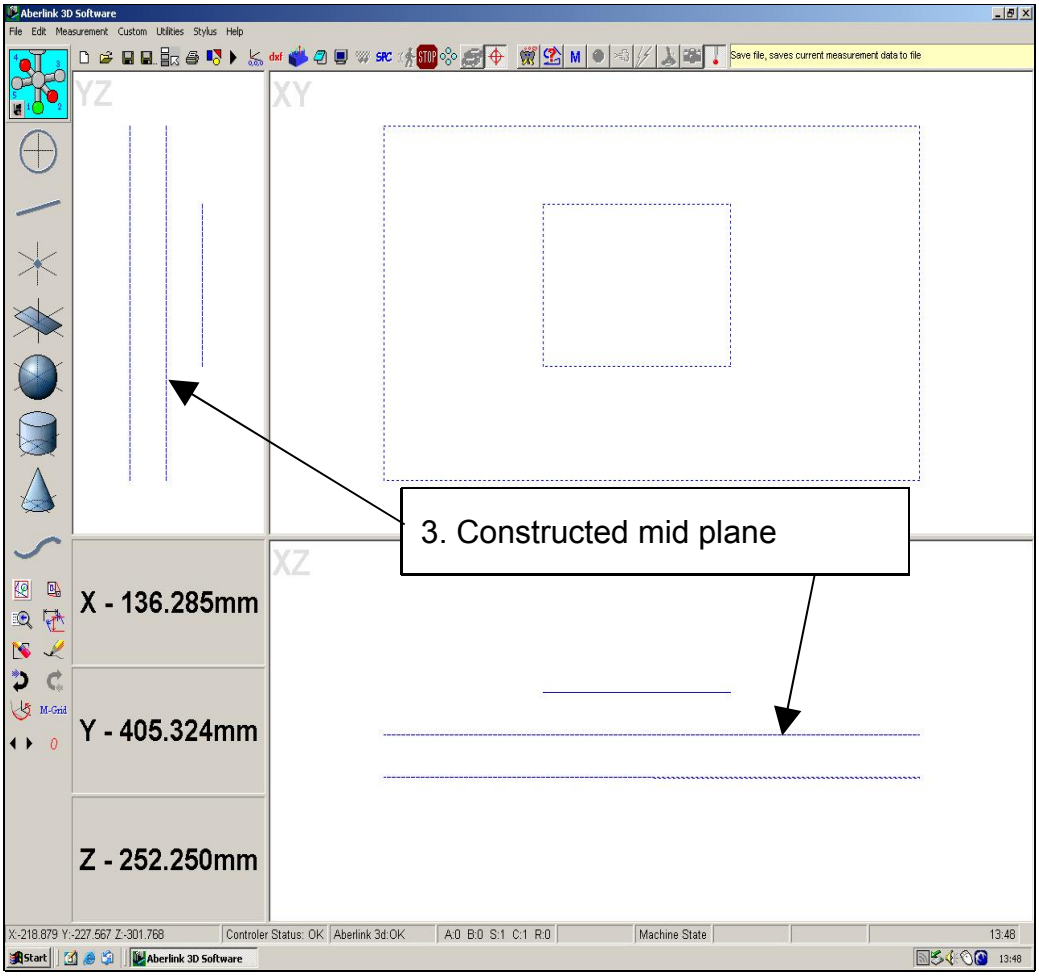


It is possible to construct a plane midway between two other planes. To do this you need to click on the ‘Plane Measure button’, from the main screen. This will bring up the Plane Measure window. Now click on the ‘Construct’ button. The Plane measure window will now shrink to a small box at the bottom of the screen:



Now select plane 1 and then plane 2 by clicking on them in any of the views. After you have done this a prompt will appear on the screen confirming you are constructing a mid-plane:

Click ‘OK’. This will bring back the Plane Measure window. OK the Plane Measure window. The screen will now return to the Main Screen, with the new plane shown equidistant between the two selected planes.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus Help
Save file, saves current measurement data to file
YZ
XY
XZ
3. Constructed mid plane
X - 136.285mm
Y - 405.324mm
Z - 252.250mm
X-218.879 Y:-227.567 Z:-301.768
Controler Status: OK
Aberlink 3d:OK
A:0 B:0 S:1 C:1 R:0
Machine State
13:48
Start
Aberlink 3D Software

4.0 THE LINE UNIT

The line unit is a very quick and accurate method of measuring flat faces, by taking 2 or more points along the length of the face, and projecting the points into a plane to produce a 2 dimensional line.

Lines may also be constructed through 2 or more previously measured points. In this case the line will have 3 dimensional properties.

Lines can also be constructed at the intersection of 2 non-parallel planes. Again this line will have 3 dimensional properties.

You can also construct a tangent line between two circles or a point and a circle

It is also possible to construct a perpendicular gauge line of a fixed length between too two other lines.

Also you can construct a line from a feature (point, circle or sphere) to a user defined end point. This can be defined either as a Cartesian or a Polar co-ordinate ie. It is possible to define the length of the line and what angle it is at.

Finally, It is also possible to construct a line that is the centre line between 2 other lines.

4.1 Measuring Lines Directly

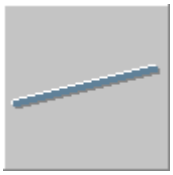
Background
When measured directly, lines are 2 dimensional features that are used to represent flat faces, which by their nature are 3 dimensional features. This is achieved by projecting the measured points on to a defined plane to produce a straight line.

The software will automatically decide which previously measured plane is most appropriate for the line, by looking at the direction of motion of the probe when the points are taken. However, it is possible for the user to define themselves which plane the line is to be projected into, by using the Plane button within the Line Measure window.

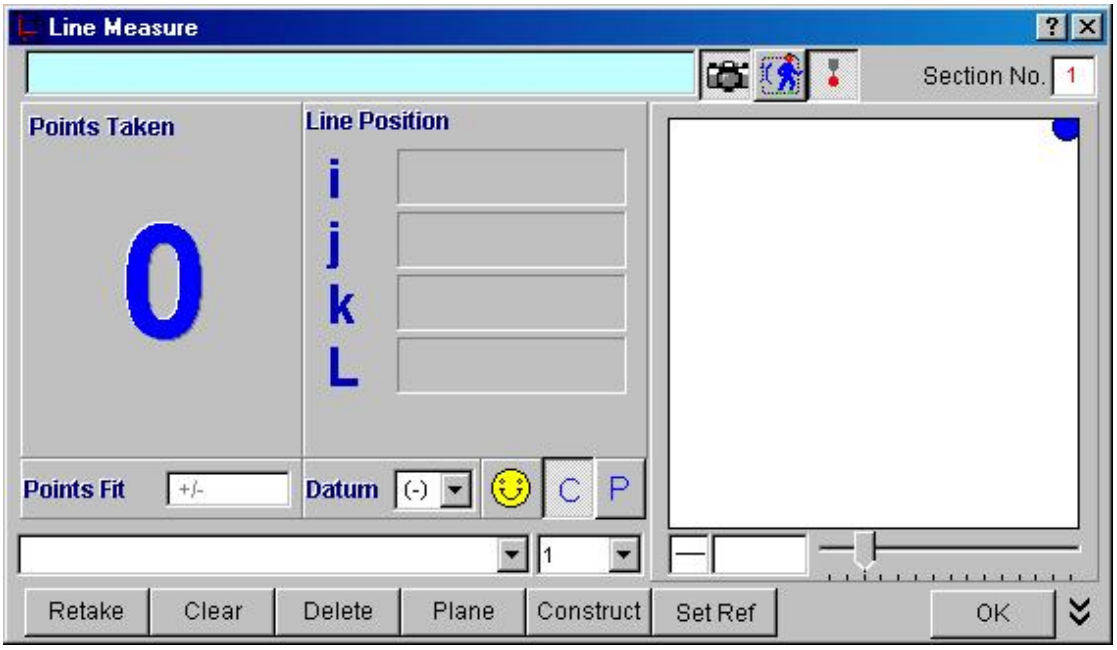
When measuring a line the software will automatically look for meaningful intersections with other lines, in order to produce a complete outline of the component being inspected. The line will extend itself automatically to meet other co-planar lines if they are close enough together. If a line has not been extended at both ends, its length will be the distance between extreme points taken, and the L value will be shown in grey. If however, a line has been successfully extended at both ends, its length is now the distance between intersections and the L value will be displayed in red.

Method

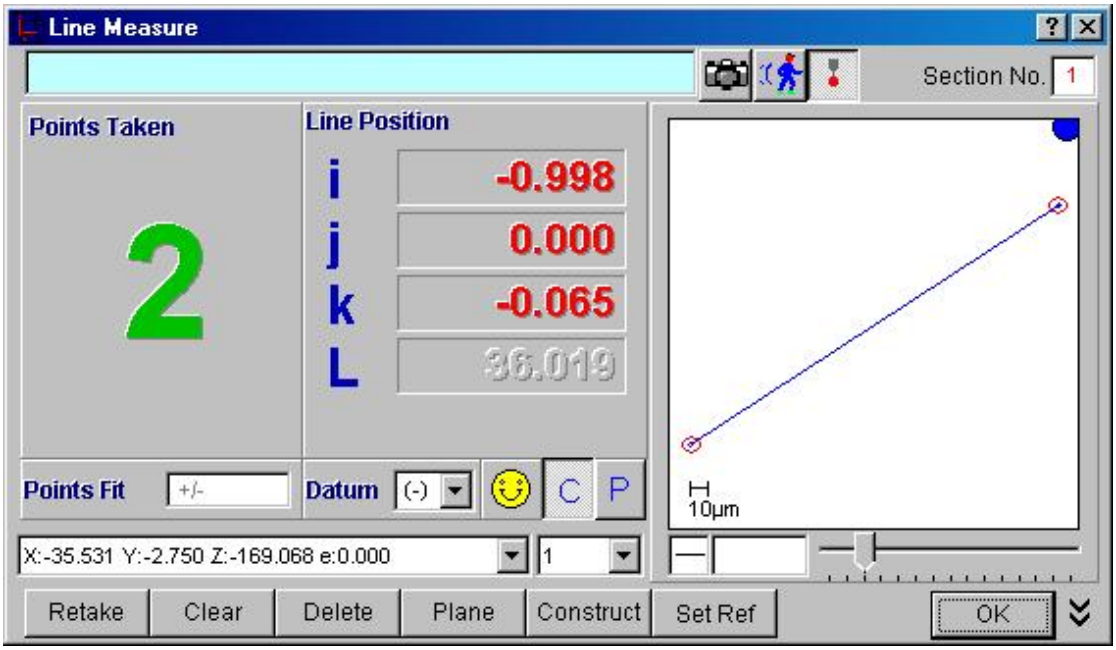
To measure a line, click on the Line Measure button, from the main screen:



This will bring up the Line Measure window:



Now take measurement points on the face that you wish to measure. After two points a pictorial representation of the line will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The I, J and K values displayed represent the direction vector for the line and the L value represents its length.

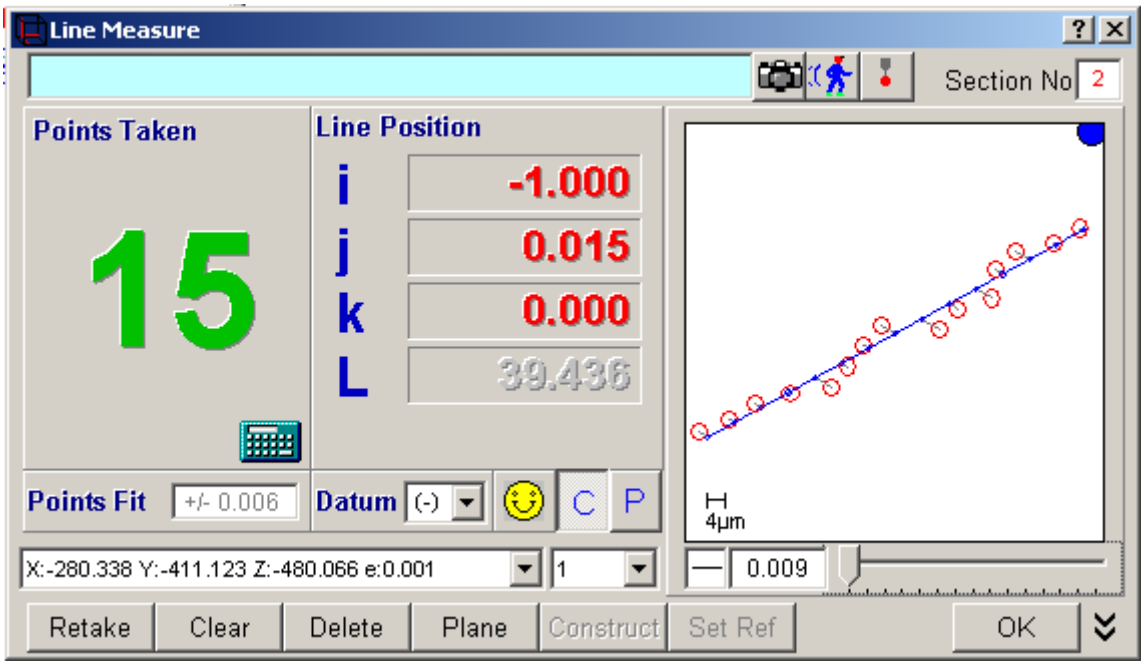


Note that if a line is measured in isolation, the length given will be the distance between the extreme measurement points and will therefore be of little relevance. In this case the L value will be shown in grey. The software does, however, automatically extend lines to look for obvious intersections with other lines. If a line has been successfully extended at both ends then the length between intersections will be displayed in red.

A line can always be calculated to fit through two points, therefore for the first two points the straightness value for the line, shown under the graphical representation, will remain blank.

It is therefore good practice to take at least one more point. On the third point the software will now be able to calculate a value for the fit of the points (3 sigma) on its calculated ‘best fit’ line and a value for the straightness of this line.

In general the more points taken on a line, the more information can be gleaned, and a minimum of three points is recommended. Similarly the points taken should cover as much of the face being measured as possible. The graphical representation of the line shows the points taken as a red circle with a blue line representing the ‘best fit’ line through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

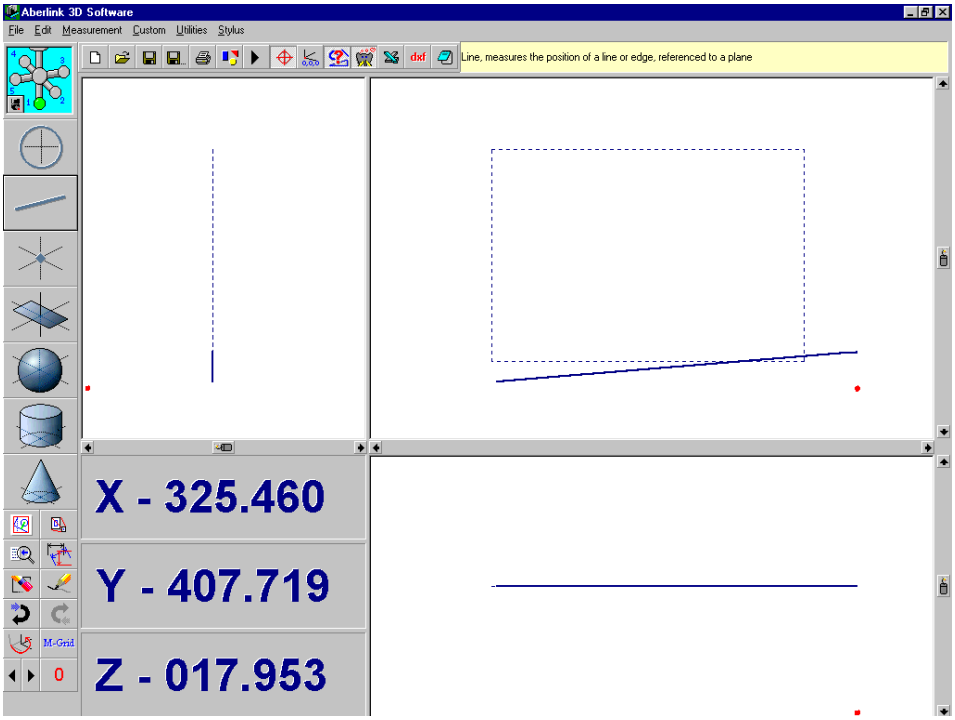


If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points using the retake and clear buttons.

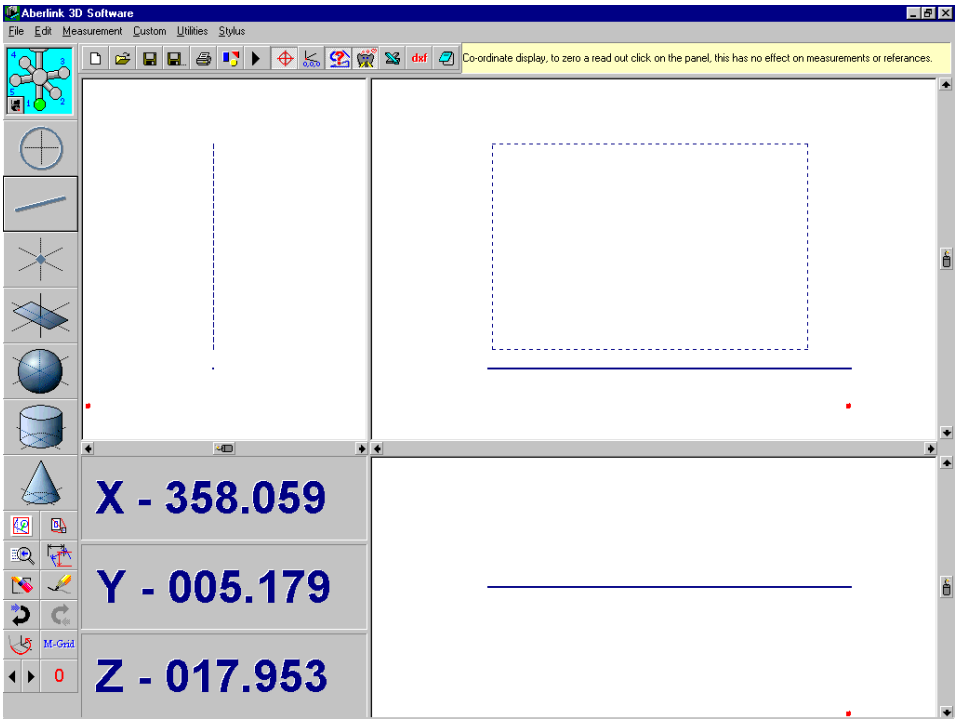
4.1.1 When to OK a line, and when to reference it.

If you OK the line, the measurement window will close, and the program will return to the main screen.

The graphical representation of the line should be shown as a solid blue line in all of the 3 views drawn at whatever angle the line has been measured at:


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Line, measures the position of a line or edge, referenced to a plane
X - 325.460
Y - 407.719
Z - 017.953
M-Grid
0

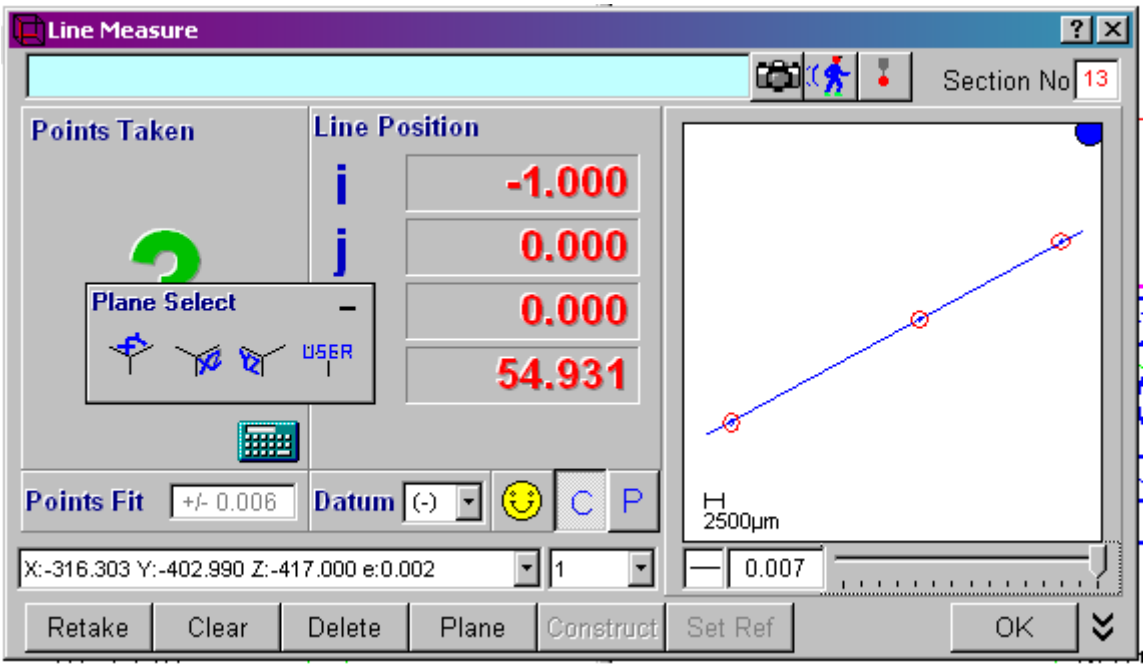
If, however, you set the line as a reference by clicking on 'Set Ref' before clicking OK then the graphical representation of the line will once again be shown as a solid blue line within the reference plane in 2 of the 3 views, but this time the graphics picture will be aligned to the line.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 358.059
Y - 005.179
Z - 017.953
M-Grid
0

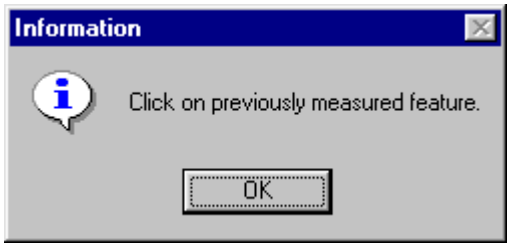
Note – If a second line is set as a reference, this will override the initial reference line, and the graphical representation will now re-align itself to the second line. The first line that was referenced will now be treated as a reference point at its mid-point, and may be used to define the position of another plane.

4.1.2 Measuring a line in a user defined plane

Rather than letting the software decide which is the most appropriate plane for a line to be projected into, it is possible for the user to define this. Having clicked on the Line Measure button to bring up the Line Measure window, click on the 'Plane' button. This will bring up a window as follows:



If you are simply defining the correct orthogonal plane, just click on the appropriate XY,XZ or YZ button that defines the line in the direction that you require, and then continue with the line measurement. If the required plane is not one of the orthogonal planes, you can still measure the line, as long as the relevant plane has previously been measured. Now click on the '**User**' button, and the following prompt will appear:



Now simply click on ‘OK’ and then click on the previously measured plane before continuing with the line measurement in the usual way.

4.2 Constructing Lines

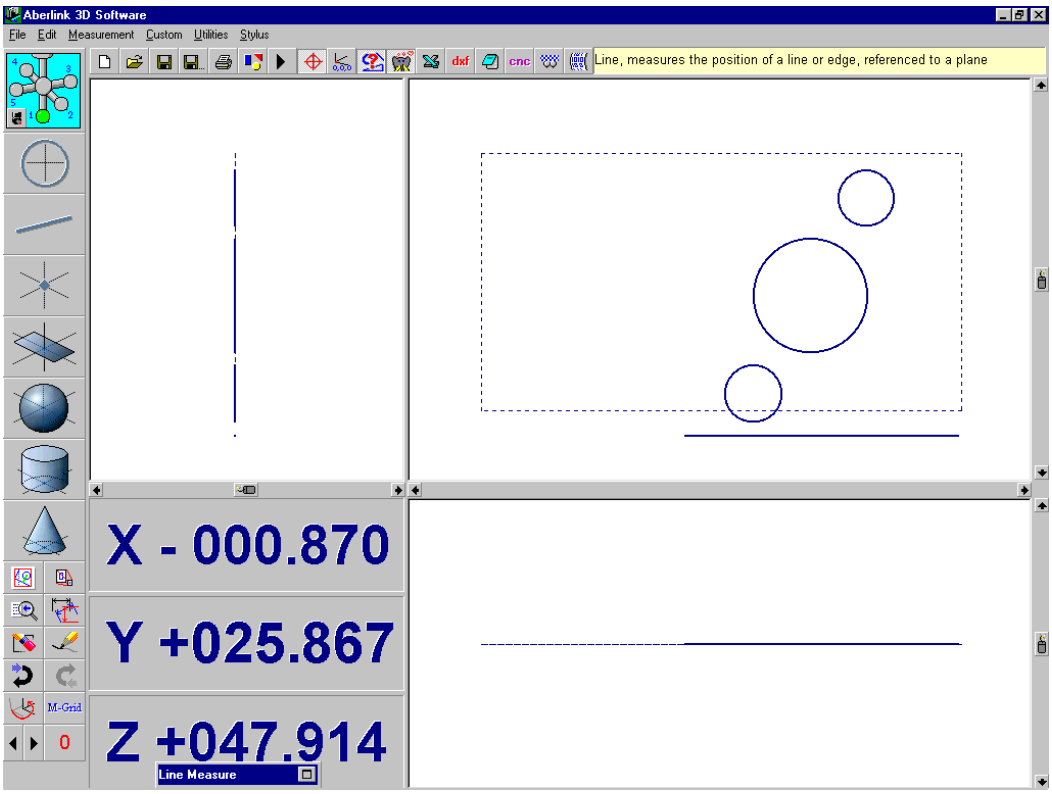
4.2.1 Constructing Lines through two or more points

Background

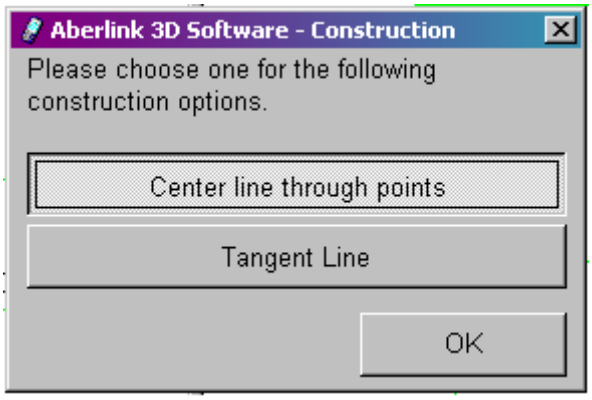
Lines can be constructed through existing points in a very similar way to measuring then directly. The only difference is that instead of fitting a line through measurement points taken with the probe, you construct it through previously measured circles, points or spheres.

Method

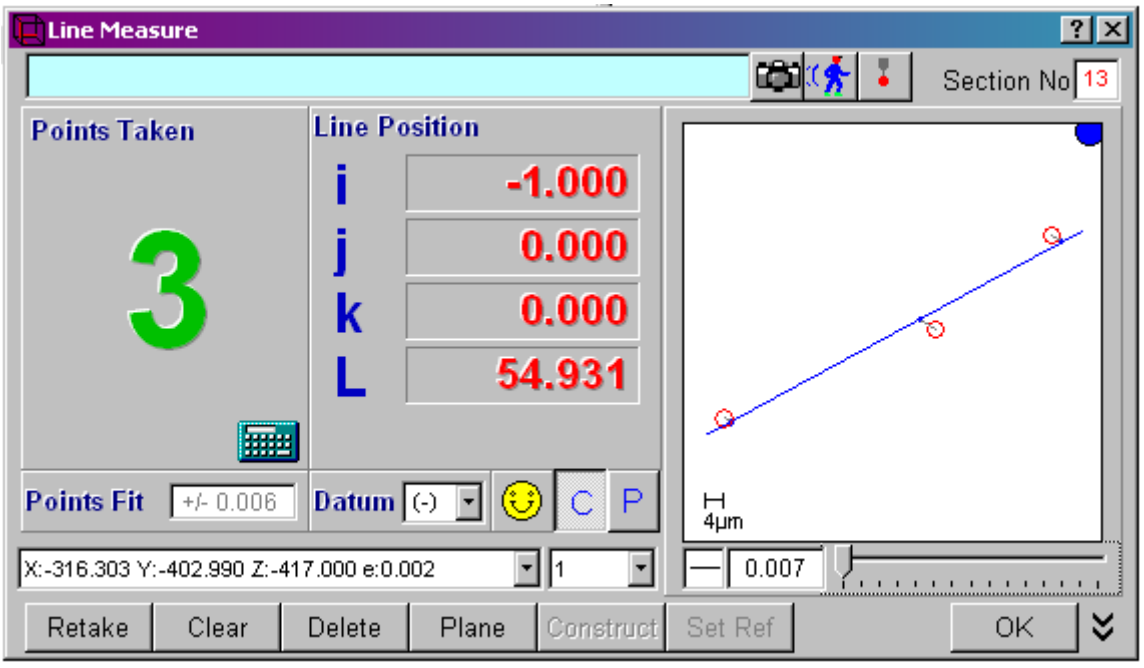
To construct a line, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the ‘Construct’ box. The Line Measure window will now shrink to a small box at the bottom of the screen.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Line, measures the position of a line or edge, referenced to a plane
X - 000.870
Y +025.867
Z +047.914
Line Measure

Now simply select the features that you wish to use to construct the line by clicking on them. After you have selected the first feature a prompt will appear, to confirm what is happening:


Aberlink 3D Software - Construction
Please choose one for the following construction options.
Center line through points
Tangent Line
OK

Choose the **'Center line through points'** option then Click on **OK**, after this continue to click on the remaining features that make up the line. When you have selected all the desired features bring the Line Measure window back by clicking on the right hand end of the shrunken box. A graphical representation of the line will appear with values for the fit and straightness given if more than 2 points were selected.

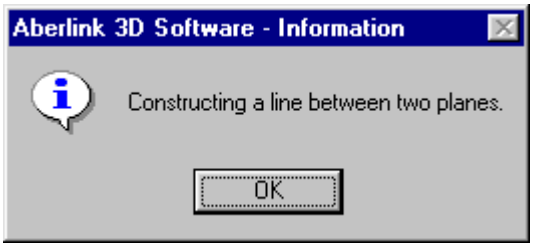


You may now click on 'OK' or 'Set Ref' as appropriate, as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

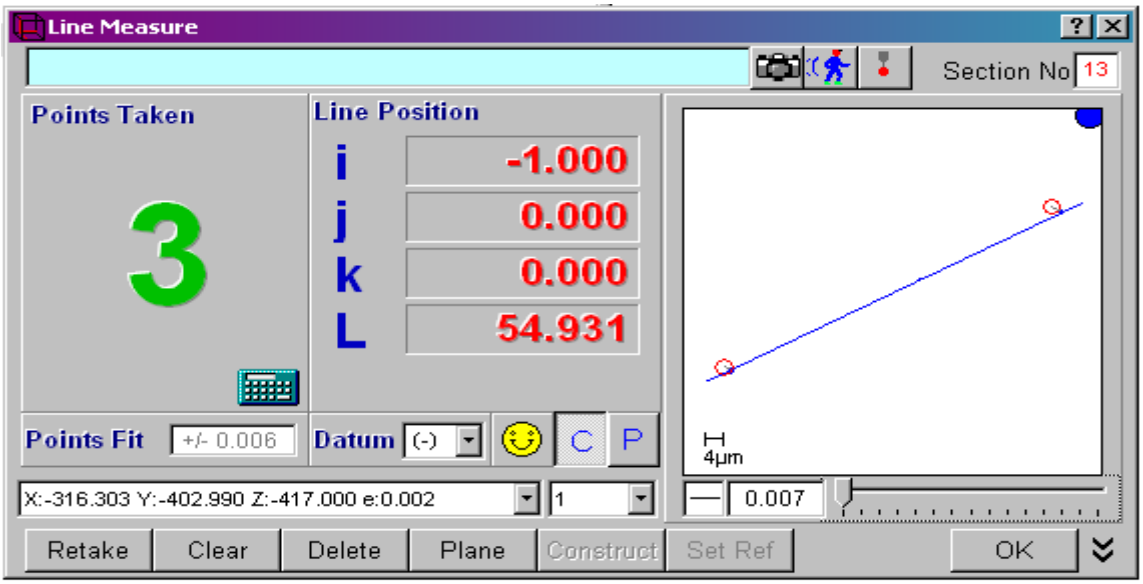
4.2.2 Constructing a Line at the Intersection of two Planes

Background
In addition to constructing lines through existing points, lines can also be constructed at the theoretical intersection of two non-parallel planes.

Method
To construct a line at the intersection of two planes, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the 'Construct' box. The Line Measure window will now shrink to a small box at the bottom of the screen. Now simply select the planes that you wish to use to construct the line by clicking on them. After you have selected the first plane a prompt will appear, to confirm what is happening:



Click on **OK**, and then click on the other plane that creates the desired line. The Line Measure window will reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:

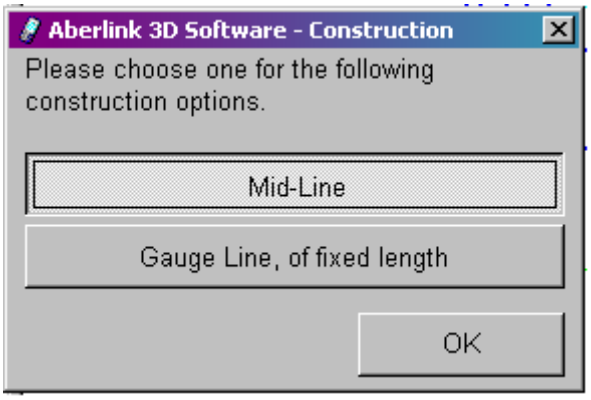


You may now click on '**OK**' as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

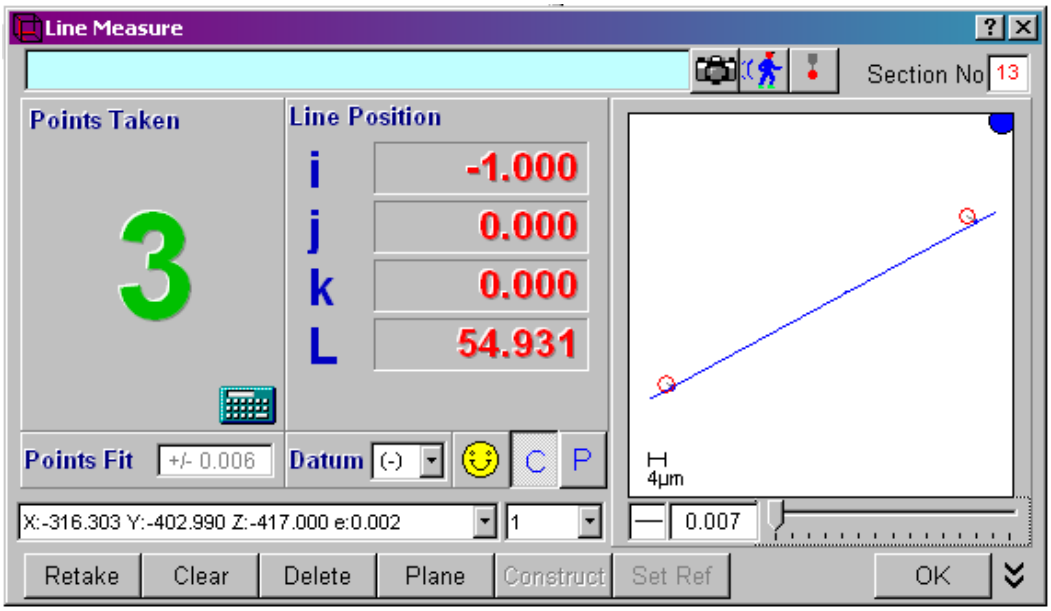
4.2.3 Constructing a Centre Line between two other Lines

Background
One other line that can be constructed is a centre line between two other lines. If the 2 lines selected are not parallel, then the centre line constructed will bisect the angle between them.

Method
To construct a centre line between 2 other lines, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the 'Construct' box. The Line Measure window will now shrink to a small box at the bottom of the screen. Now simply select the lines that you wish to use to construct the centre line by clicking on them. After you have selected the first line a prompt will appear, to confirm what is happening:


Aberlink 3D Software - Construction
Please choose one for the following construction options.
Mid-Line
Gauge Line, of fixed length
OK

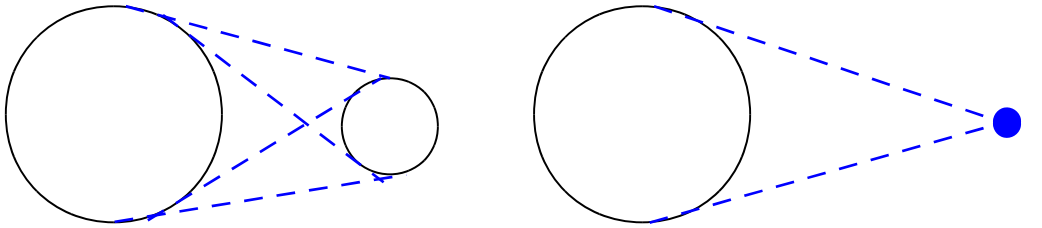
Choose the **'Mid-line'** option then Click on OK, and after this click on the other line that creates the desired centre line. The Line Measure window will reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:



You may now click on ‘OK’ as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

4.2.4 Constructing a Tangent Line between 2 circles or a point & circle

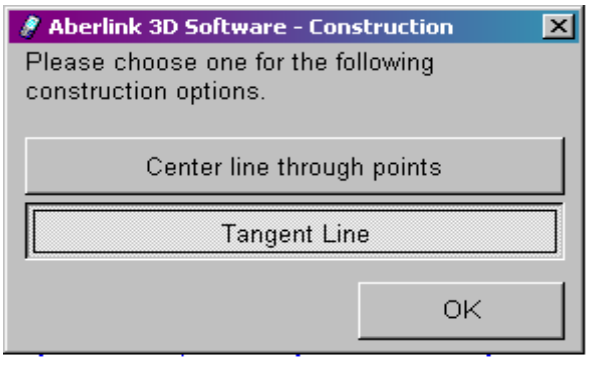
Background
With this function you can construct tangential lines between two circles or between a point and a circle.



Tangent lines between two Circles

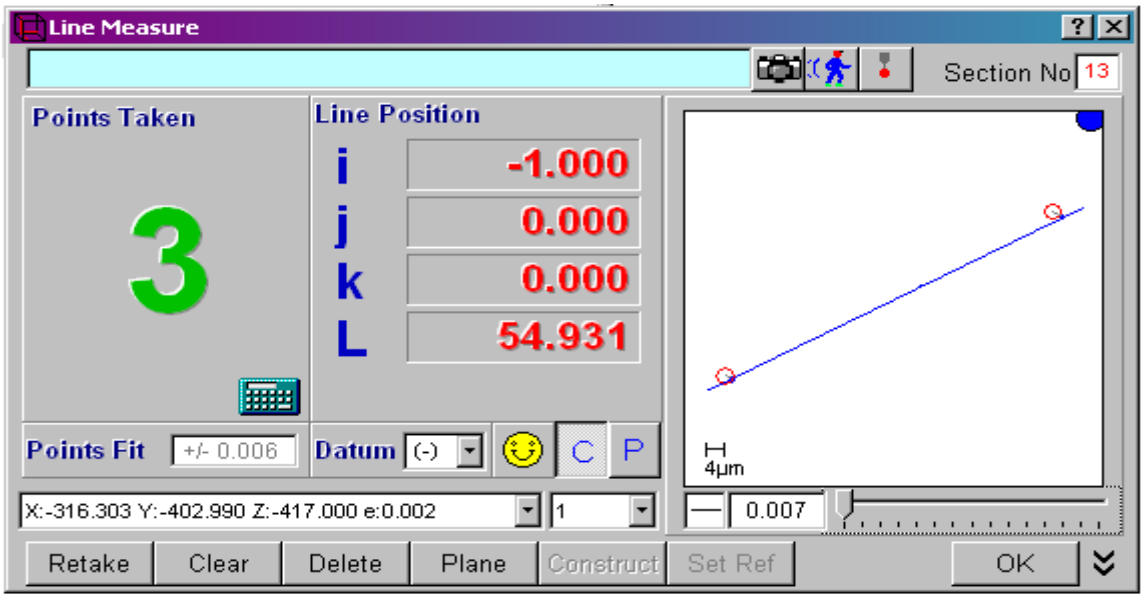
Tangent lines between a Point and a Circle

Method
To construct a tangential line between two circles or between a point and a circle, Click on the ‘Line Measure button’, from the main screen. This will bring up the Line Measure window. Now click on the ‘Construct’ box. The Line Measure window will now shrink to a small box at the bottom of the screen. Note that between 2 circles there are actually 4 possible tangent lines that can be constructed, and 2 possible tangent lines between a point and a circle. Now simply select the first circle (or the point) followed by the second circle being careful to click on it at the position closest to where the desired tangent line will pass. The following prompt will then appear:


Aberlink 3D Software - Construction
Please choose one for the following construction options.
Center line through points
Tangent Line
OK

Choose the **‘**Tangent line**’** option then Click on ‘OK’.

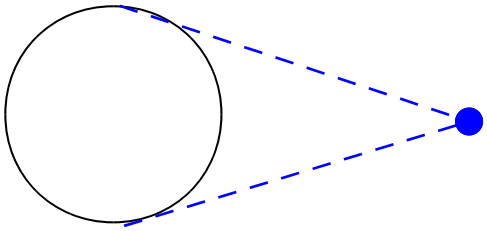
The Line Measure window will now reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:



You may now click on 'OK' as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

Example 1: <u>constructing a tangent line between a point and a circle.</u>

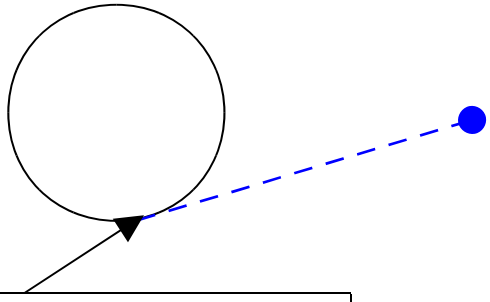
Note that there are 2 possible options, as follows:



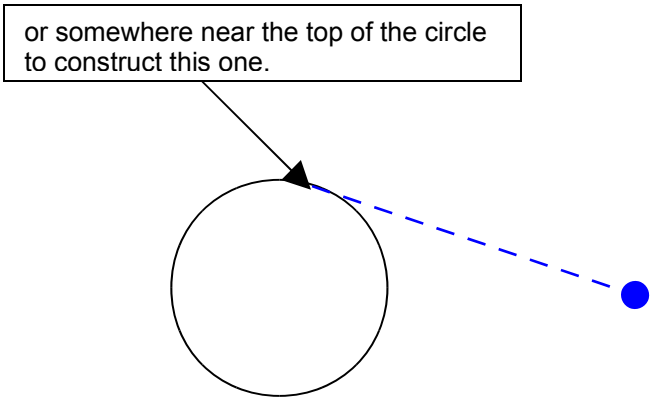
1) Open a Line Measure Window and click on 'Construct' so that the window shrinks to a small box at the bottom of the screen.

2) Click on the Point to select it.

3) Now click on the circle at the appropriate position to choose the desired tangent line.

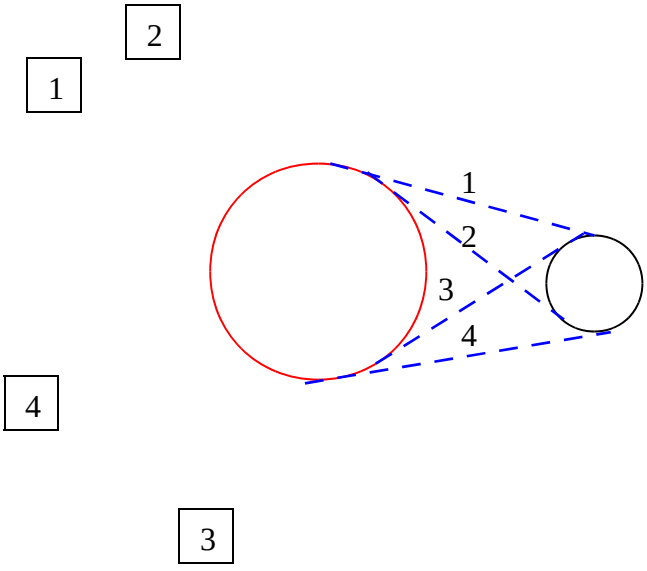


Click somewhere near the bottom of the circle to construct the tangent line



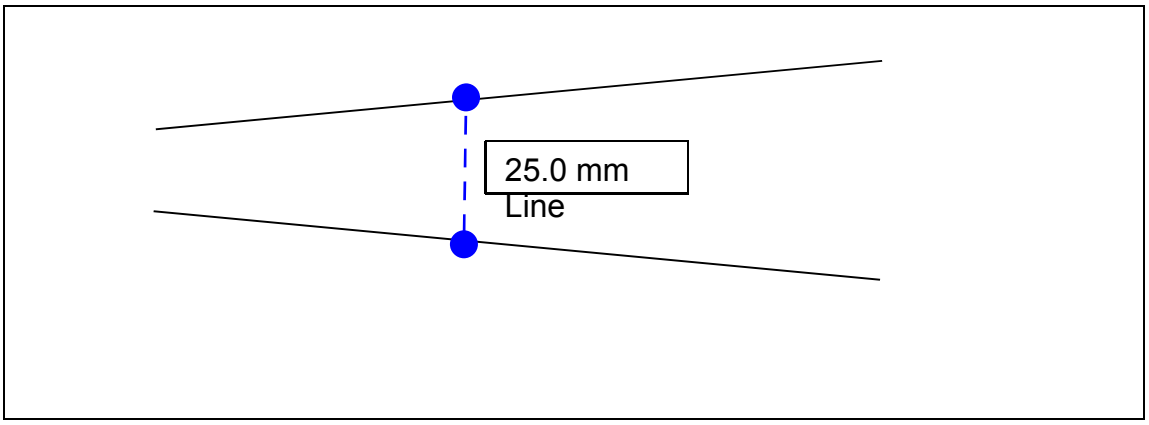
4) OK the Line Measure Window to return to the Main Screen.

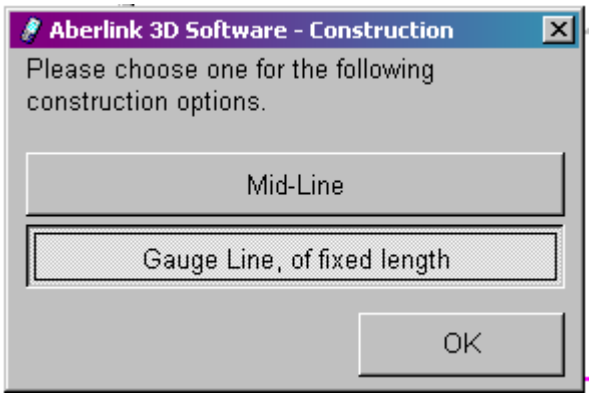
Tip : Often the tangent lines between 2 circles can touch the circles at a very similar position making it difficult to choose the correct line required. In this case highlight the 2nd circle using the Feature Select buttons and click at a position that will clearly define the desired tangent line. For example, if you select the larger circle second, using the Feature Select function you can now click high up on the box labelled '2' in order to construct tangent line no2. Note that this position is through where tangent line no.2 will pass, and is clearly closer to no.2 than any of the others. Similarly the other boxes also show possible positions to click that will uniquely define the other tangent lines.



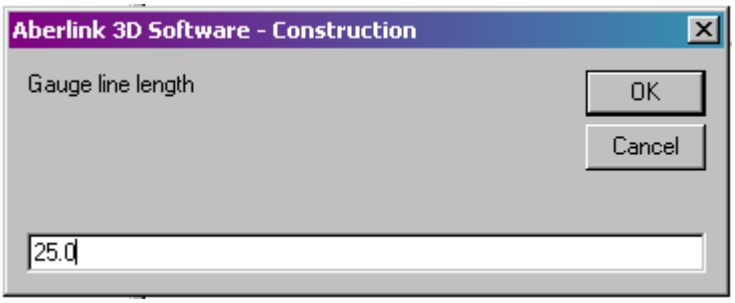
4.2.5 Constructing a Gauge Line of Fixed Length between two other Lines

Background
This function lets you construct a gauge perpendicular line of a fixed length between two other lines set at an angle to each other.

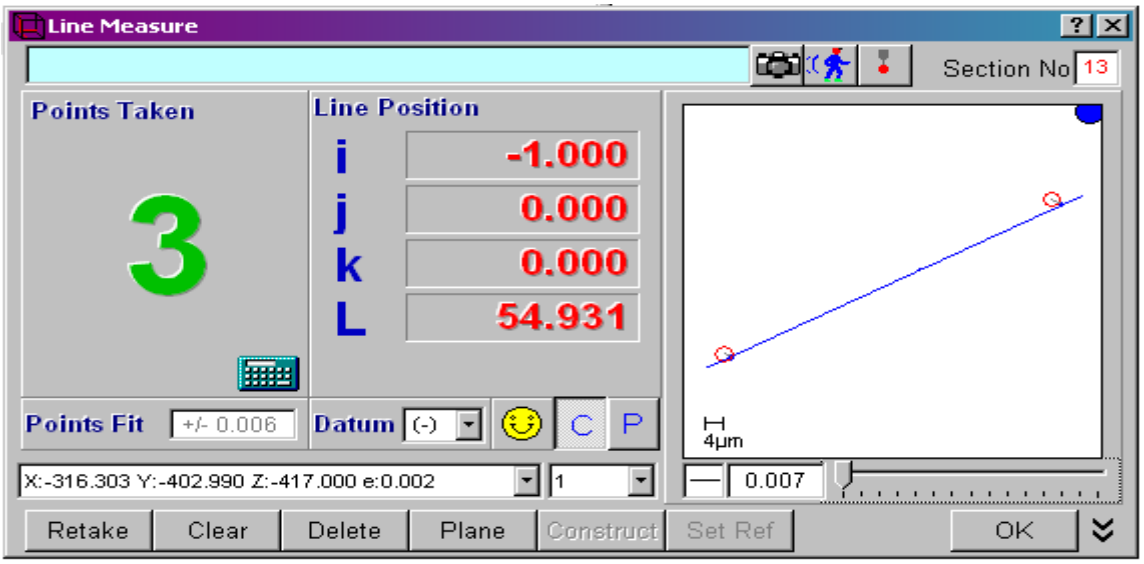



Aberlink 3D Software - Construction
Please choose one for the following construction options.
Mid-Line
Gauge Line, of fixed length
OK

Choose the '**Gauge line of fixed length'** option then Click on '**OK**', and after this click on the other line that you wish to connect the gauge line too. A prompt window will appear asking you for the length of the gauge line:



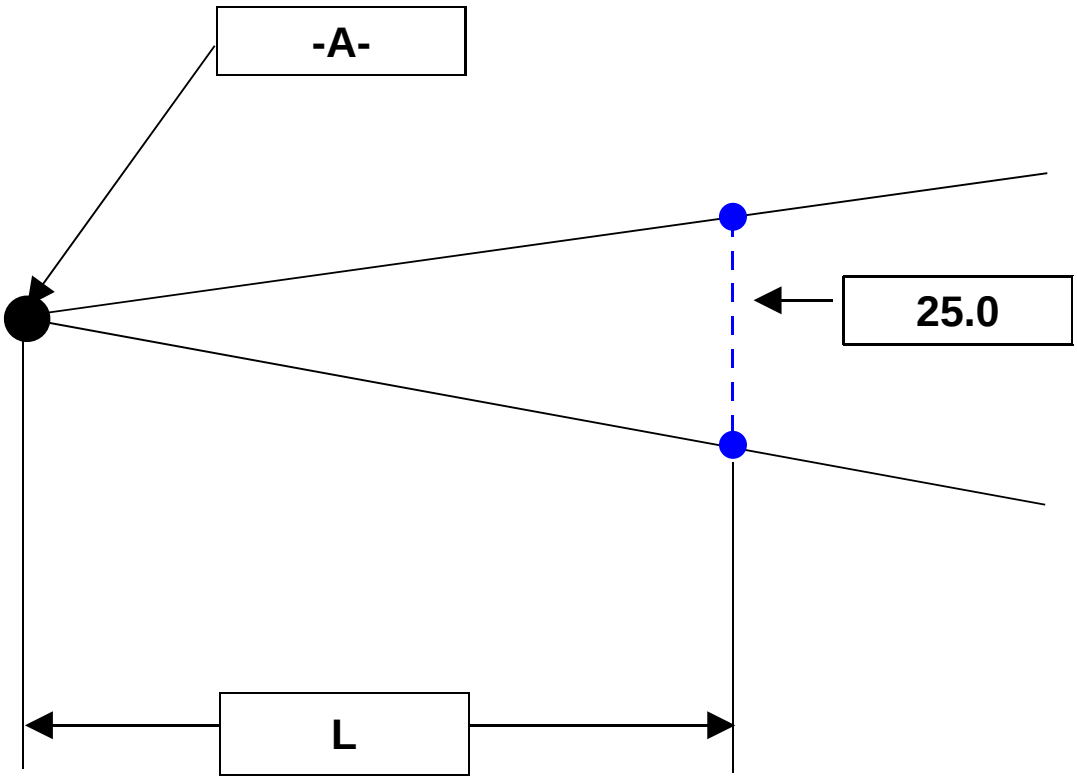
Enter the size of the gauge and click '**OK**' . The Line Measure window will now reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:



Now click on 'OK' The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

Example

You have measure two lines on opposite sides of a cone and you wish to know how far from the intersection is the point were the two lines are precisely 25mm apart



5.0 THE CIRCLE UNIT

The circle unit is a very quick and accurate method of directly measuring holes, round bosses or arcs by taking 3 or more measurement points around the circumference.

Circles can also be constructed through 3 or more previously measured circles or points, for instance to create a pitch circle diameter.

Tangential circles of a given size can also be constructed between 2 lines or 2 circles.

In addition, circles can be constructed at the intersection of 3-dimensional features, such as the intersection of a plane with a perpendicular cylinder or cone.

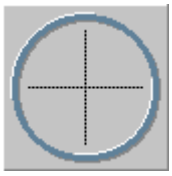
5.1 Measuring Circles Directly

Background
Circles are 2 dimensional features that are used to measure holes, bosses or arcs, which by their nature are 3 dimensional features. This is achieved by projecting the measured points on to a defined plane to produce a 2 dimensional circle.

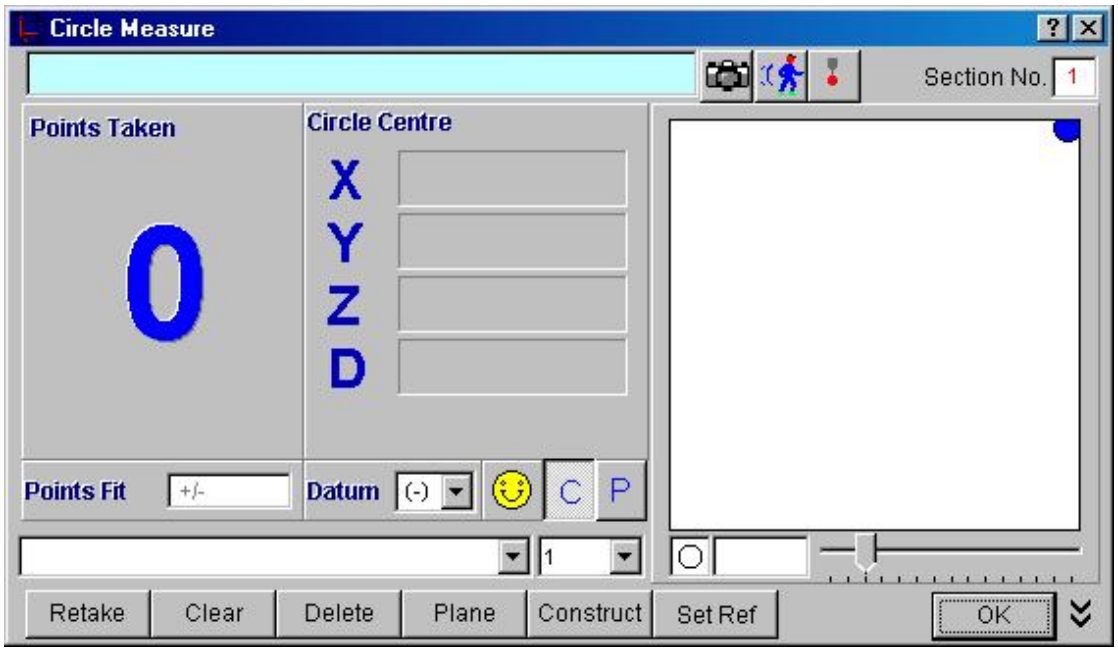
The software will decide the most appropriate plane for a circle automatically, by looking at the direction of motion of the probe when the points are taken

If the circular feature consists of less than 180 degrees, then the software will draw it as an arc between the extreme points. As soon as the measurement points span more than 180 degrees then a full circle will be described.

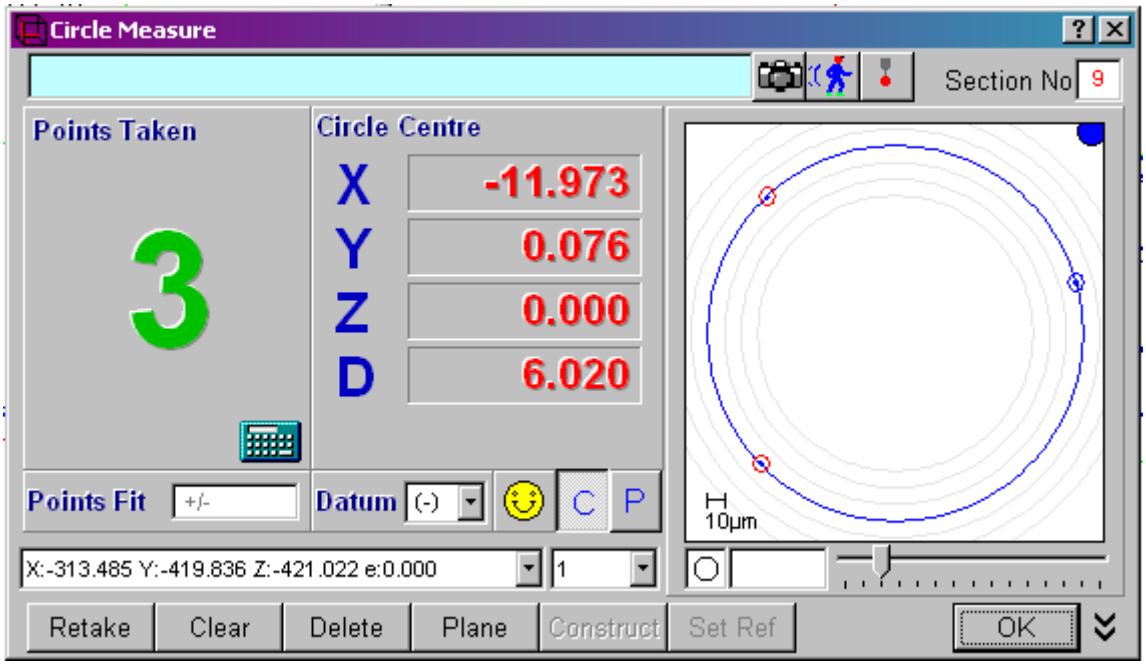
Method
To measure a circle, click on the Circle Measure button, from the main screen:



This will bring up the Circle Measure window:



Now take measurement points on the hole or boss that you wish to measure. After 3 points a pictorial representation of the circle will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The X,Y and Z values displayed represent the centre of the circle and the D value represents its diameter.

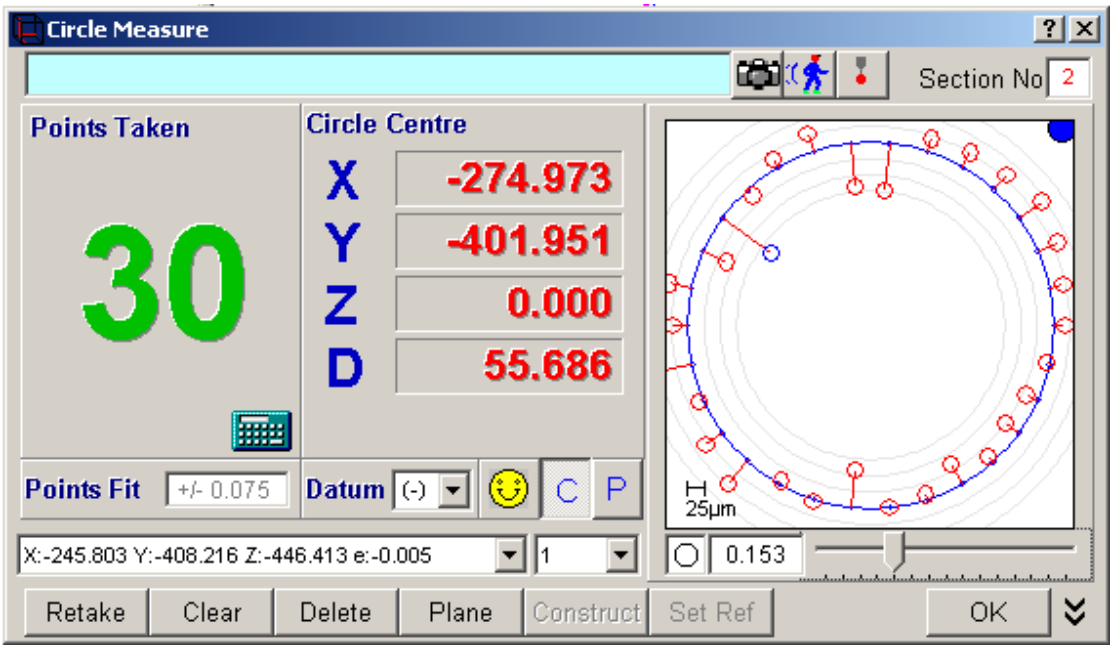


A circle can always be calculated to fit perfectly through three points, therefore for the first 3 points the roundness value for the circle, shown under the graphical representation, will remain blank.

It is therefore good practise to take at least one more point. On the fourth point the software will now be able to calculate a value for the fit of the points (2-sigma) on its calculated best fit circle and a value for the roundness of this circle

In general the more points taken on a circle, the more information can be gleaned, and a minimum of 4 points is recommended. Similarly the points taken should cover as much of the circumference of the circle as possible.

The graphical representation of the circle shows the points taken as a red circle with a blue line representing the best fit circle through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.



If you have accidentally probed on the wrong place on the work piece, or are unhappy with a point, you can always erase either an individual point or all the stored points using the retake or clear buttons.

5.1.1 When to OK a Circle, and when to reference it.

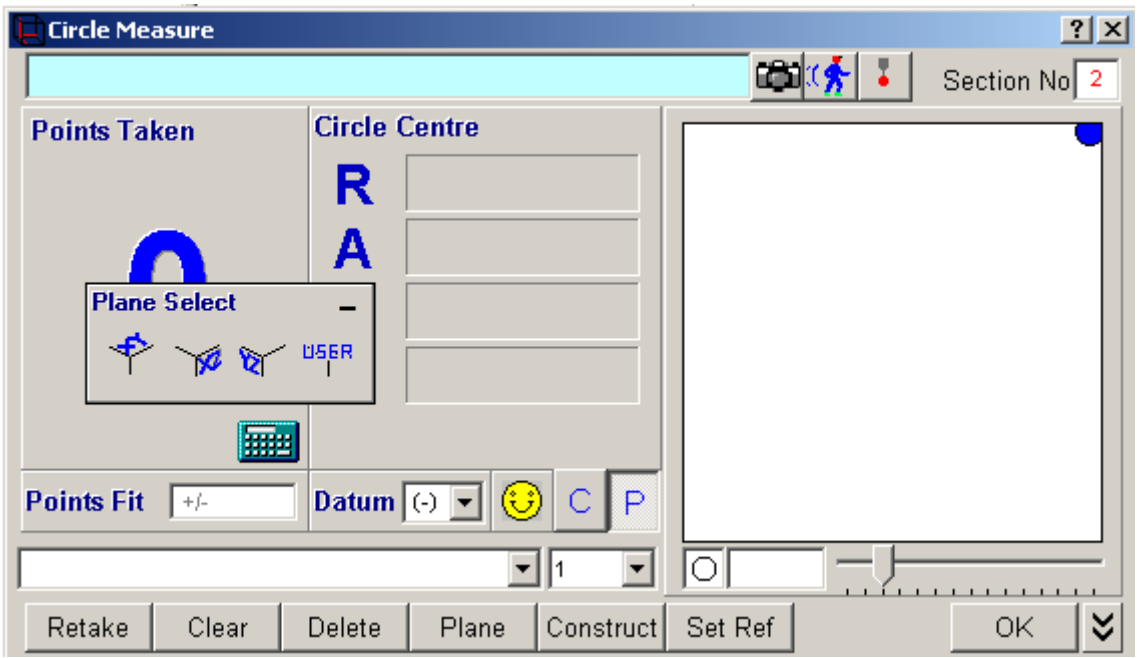
If you select 'OK' the Circle Measure window will close and the centre co-ordinates of the circle will be relative to any references previously set, (or if none have been set then to the machine's own reference point).

If however, you click Set Ref before OK, then the centre of the circle will be set to 0,0,0.

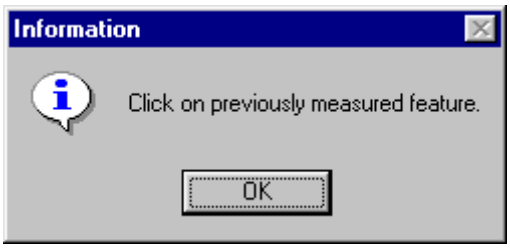
> ***Note*** *– Only one circle can ever be set as a reference at any point in time. If a second circle (or other point) is set as a reference, this will override the initial reference point, and the centre co-ordinates of the second circle or point will be set to 0,0,0.*

5.1.2 Measuring a circle in a user defined plane

Rather than letting the software decide which is the most appropriate plane for a circle to be projected into, it is possible for the user to define this. Having clicked on the Circle Measure button to bring up the Circle Measure window, click on the 'Plane' button. This will bring up a window as follows:



If you are simply defining the correct orthogonal plane, just click on the appropriate XY,XZ or YZ button, and then continue with the circle measurement. If the required plane is not one of the orthogonal planes, you can still measure the circle, as long as the relevant plane has previously been measured. Now click on the 'User' button, and the following prompt will appear:



Now simply click on 'OK' and then click on the previously measured plane before continuing with the circle measurement in the usual way.

5.2 Constructing Circles

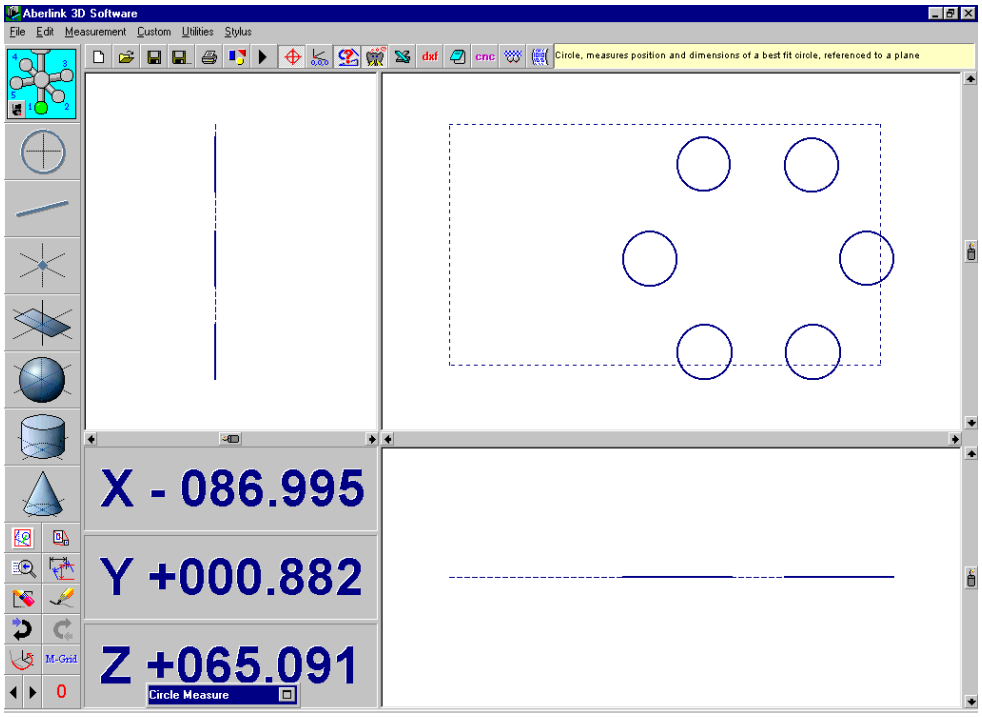
5.2.1 Constructing a Pitch Circle Diameter

Background

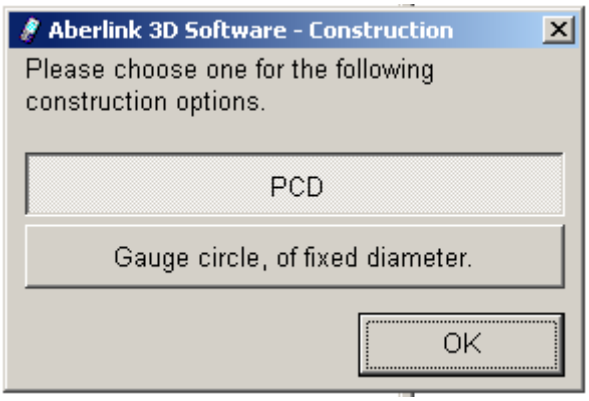
Pitch circle diameters can be inspected in a very similar way to measuring a circle directly. The only difference is that instead of fitting a circle through measurement points taken with the probe, you construct a circle through previously measured circles or points.

Method

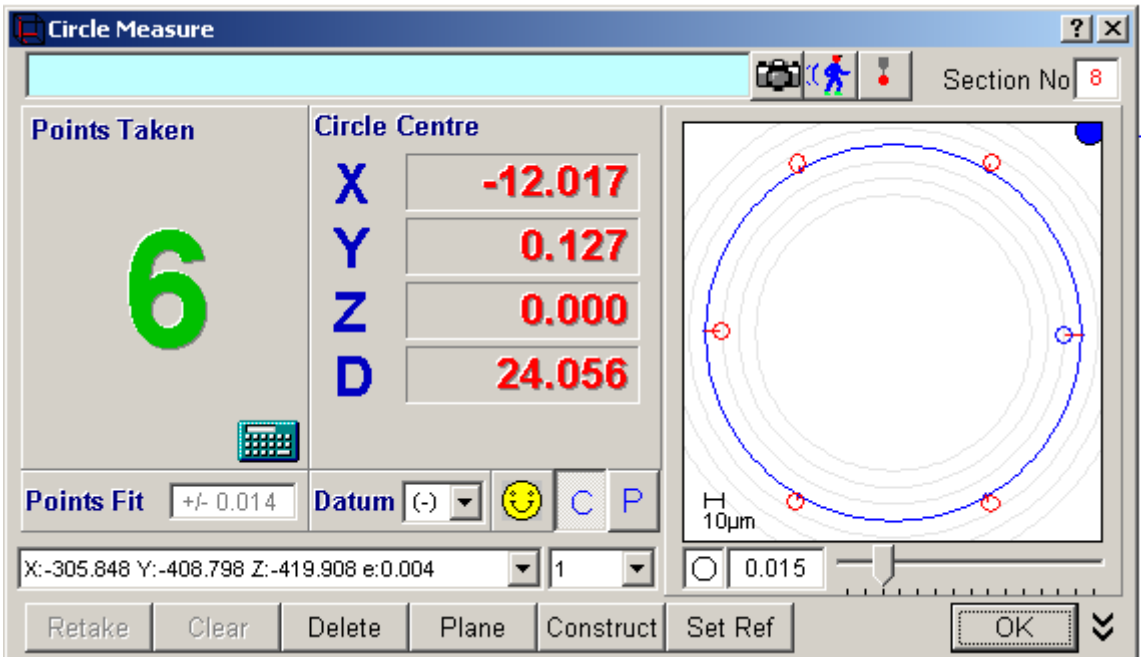
To construct a pitch circle diameter, click on the Circle Measure button, from the main screen. This will bring up the Circle Measure window. Now click on the 'Construct' box. The Circle measure window will now shrink to a small box at the bottom of the screen.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Circle, measures position and dimensions of a best fit circle, referenced to a plane
X - 086.995
Y +000.882
Z +065.091
Circle Measure

Now simply select the features that you wish to use to construct the circle by clicking on them. After you have selected the first feature a prompt will appear, to confirm what is happening:


Aberlink 3D Software - Construction
Please choose one for the following construction options.
PCD
Gauge circle, of fixed diameter.
OK

Select the PCD option then click on Yes, and then continue to click on the remaining features that make up the PCD. When you have selected all the desired features bring the Circle measure window back by clicking on the right hand end of the shrunken box. A graphical representation of the circle will appear with values for the fit and roundness given if more than 3 points were selected.



You close the window by click on ‘OK’ button, as for any measured circle. The screen will then return to the Main Screen, and the circle will form a part of the graphical representation, as would any measured circle.

5.2.2 Constructing a Circle at the Intersection of two Features

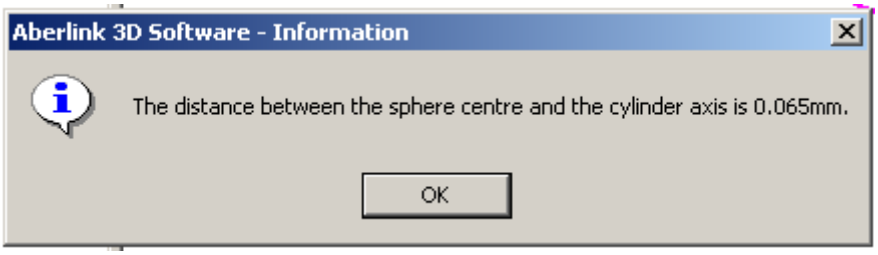
Background

In addition to constructing pitch circle diameters, circles can also be constructed at the intersection of two 3-dimensional features, such as a plane and a perpendicular cylinder. In fact circles may be constructed when any of the following features overlap:

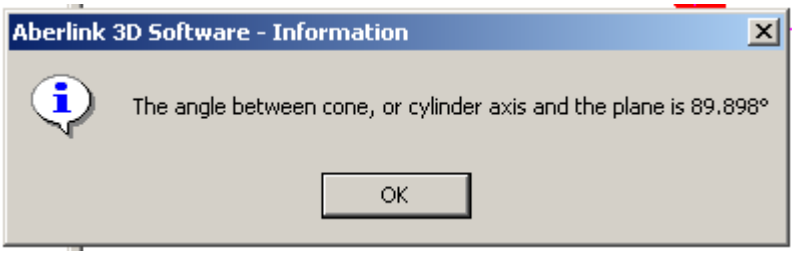
Plane/Cylinder	(if perpendicular)
Plane/Cone	(if perpendicular)
Plane/Sphere	
Cylinder/Sphere	(if on common axis)
Cone/Sphere	(if on common axis)
Sphere/Sphere	(if on common axis)
Cone/Cylinder	(if on common axis)
Cone/Cone	(if on common axis)

Method

To construct a circle at the intersection of two features, click on the Circle Measure button, from the main screen. This will bring up the Circle Measure window. Now click on the 'Construct' box. The Circle measure window will now shrink to a small box at the bottom of the screen. Now simply select the features that you wish to use to construct the circle by clicking on them. After you have selected the first feature one of two prompts will appear, to confirm what is happening, for instance:

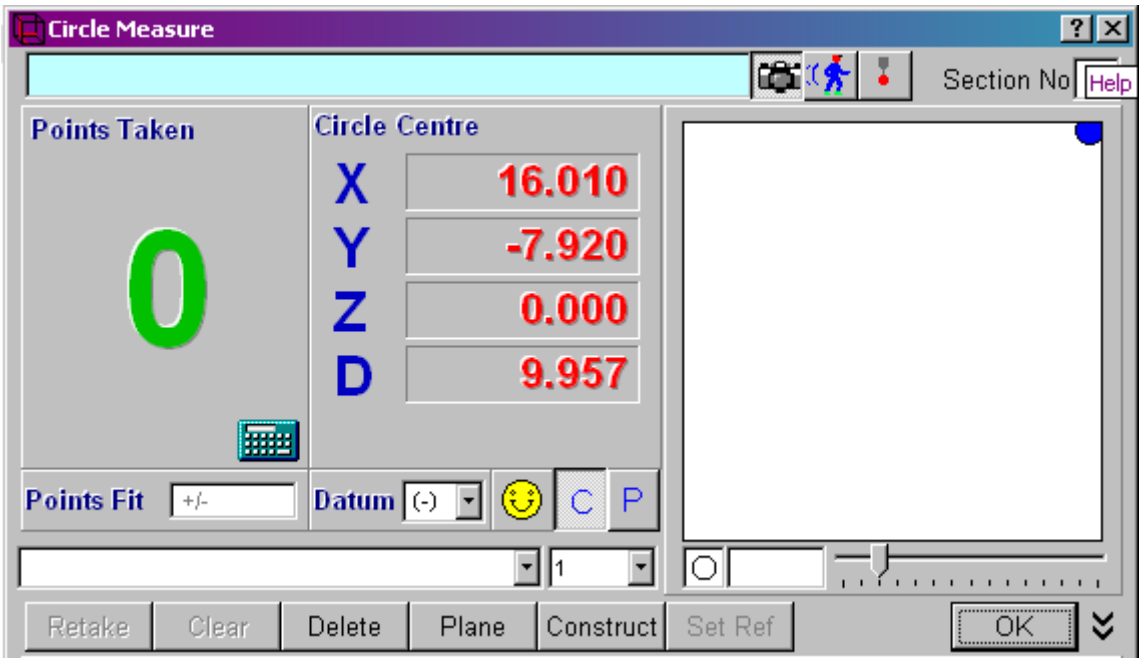


Click on OK, and then click on the other feature that creates the desired circle. If you are creating the intersection of a cylinder or cone with a plane, a different prompt will appear, telling you the angle between the cylinder or cone and the plane



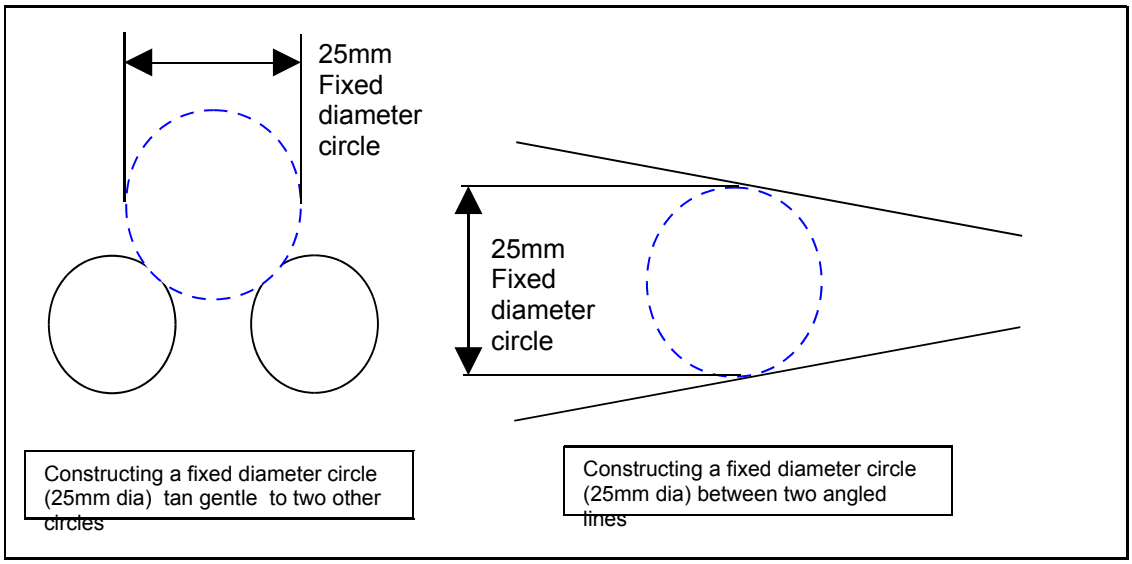
This is because if the features are not truly perpendicular, their intersection will not be a perfect circle, and this information allows the user to make a judgement of the validity of the circle that they are constructing. Now click 'OK'.

The Circle Measure window will reappear. The number of points taken will be 0, and the graphics part of the window will be blank:



You may now click on 'OK' or 'Set Ref' as appropriate, as for any measured circle. The screen will then return to the Main Screen, and the circle will form a part of the graphical representation.

8.2.3 Constructing of gauge circle of a fixed diameter between two lines

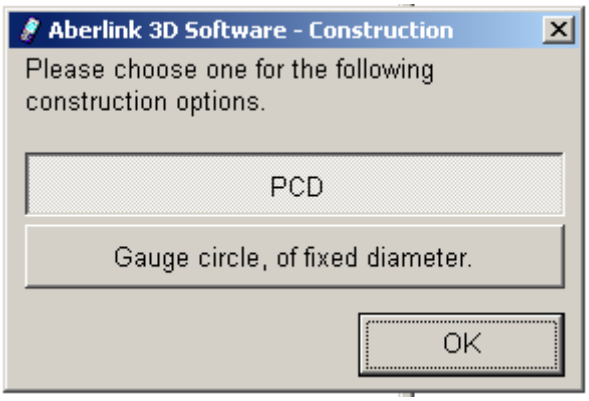


It is possible to construct a tangential circle of fixed diameter between two previously measured circles or non parallel lines.

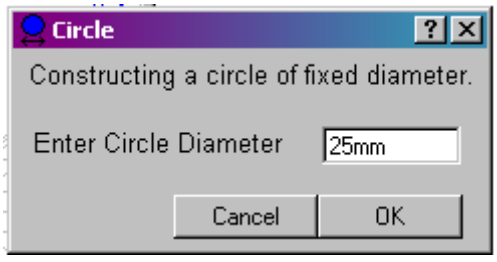
Method

To construct a fixed diameter circle between two lines or circles, click on the Circle Measure button, from the main screen. This will bring up the Circle Measure window. Now click on the 'Construct' box. The Circle measure window will now shrink to a small box at the bottom of the screen.

Now simply select the first feature that you wish to use by clicking on it followed by the second feature. The following prompt will appear, to ask what type of circle you wish to construct:


Aberlink 3D Software - Construction
Please choose one for the following construction options.
PCD
Gauge circle, of fixed diameter.
OK

Select the 'Gauge circle, of fixed diameter' option then click OK. A second prompt window will appear asking for the diameter of the gauge circle



Enter the diameter of the gauge circle to be fitted between the two features, and then click OK. This will bring the Circle measure window. Click 'OK' and the software will then return to the Main Screen, with the constructed circle shown on the graphical representation.

6.0 THE POINT UNIT

Points are an infinitesimally small position in space, which will have an X,Y and Z co-ordinate, and can be measured directly using the probe.

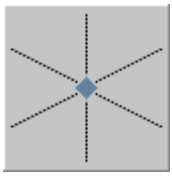
Points can also be constructed at the intersections of various features, or at the mid-point of a line.

Points may also be constructed at co-ordinates that can be entered via the keyboard.

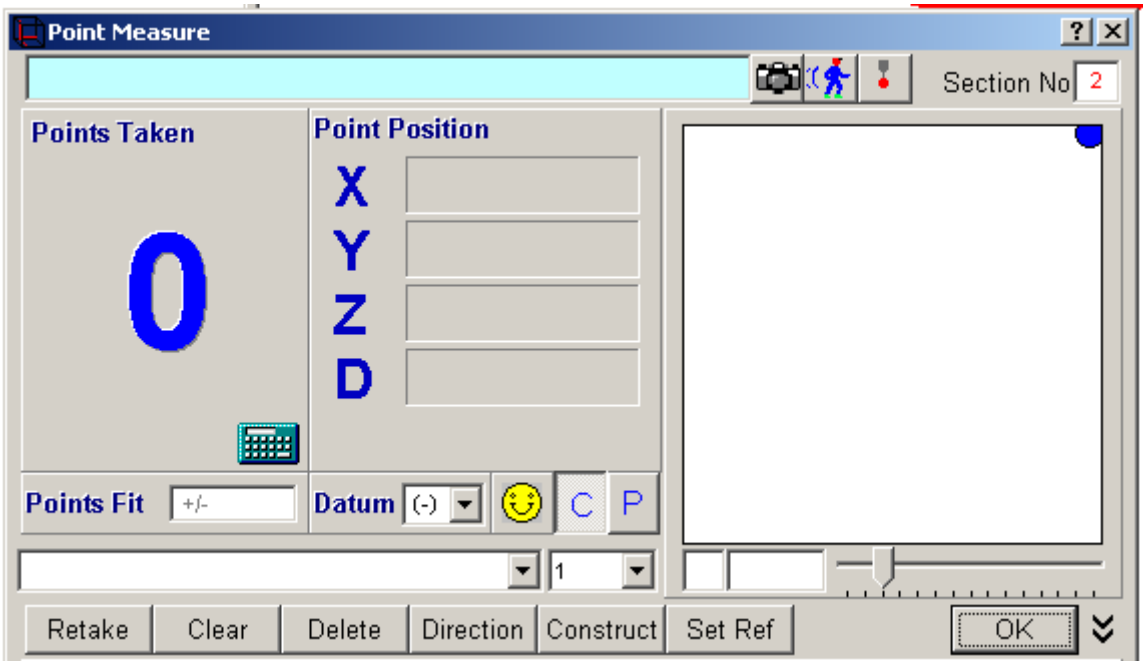
6.1 Measuring Points Directly

Background
Measuring a point directly will give a single point co-ordinate on the surface of the component. The software will automatically apply the stylus radius offset, by looking at the direction of motion of the probe when the measurement point is taken, providing that there is a previously measured feature (plane or line) to determine the direction for compensation.

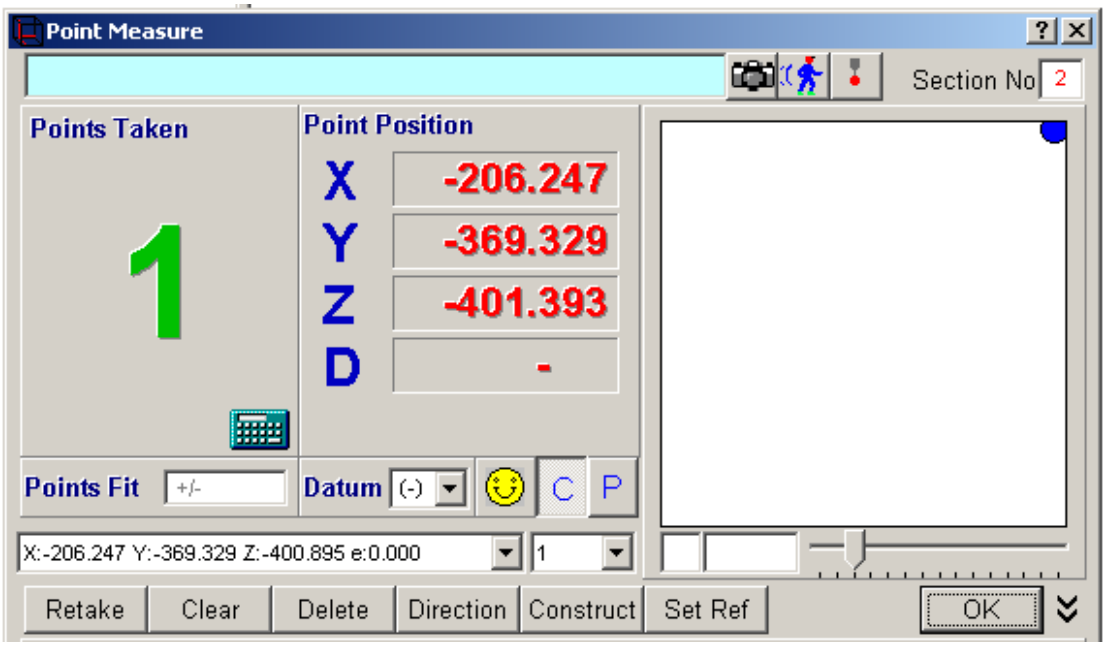
Method
To measure a point, click on the Point Measure button, from the main screen:



This will bring up the Point Measure window:

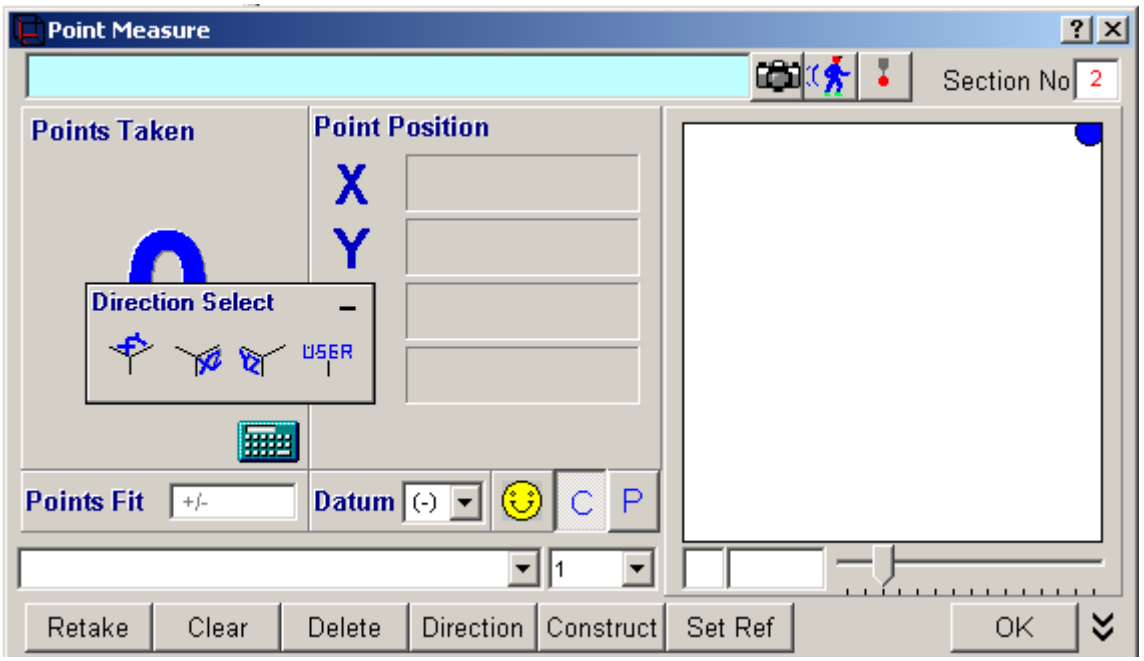


Now take a point at the position that you require. If the software is able to determine what direction to compensate for the radius of the stylus, then the point will appear in the window as follows:

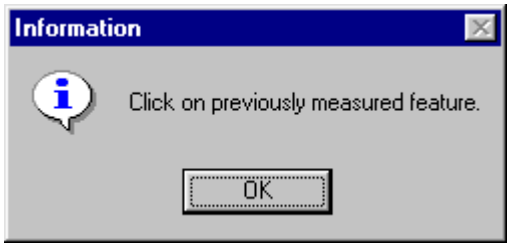


However, if it is unable to determine an appropriate direction to compensate then the following prompt will appear:

You will therefore need to click on the ‘Direction ‘button as follows:



You can now choose to compensate into an orthogonal plane by clicking on the appropriate XY,XZ or YZ button, and then continue with the measured point. If the required direction is not one of the orthogonal planes, you can still measure the point, as long as the relevant direction has been defined by a previously measured feature. In this case click on the ‘User’ button, and the following prompt will appear:



Now simply click on ‘OK’ and then click on the previously measured feature before continuing with the point measurement in the usual way.

If you have accidentally probed on the wrong place on the workpiece, or are unhappy with the point, simply take the point again. As the point

measurement function is only ever a single point, this will override the initial point taken.

6.1.1 When to OK a Point, and when to reference it.

If you select 'OK' the Point Measure window will close and the co-ordinates of the Point will be relative to any references previously set, (or if none have been set then to the machine's own reference point).

If however, you click Set Ref before OK, then the co-ordinates of the point will be set to 0,0,0.

> ***Note*** *– Only one point can ever be set as a reference at any time. If a second point (or circle or sphere) is set as a reference, this will override the initial reference point, and the centre co-ordinates of the second point (or circle or sphere) will be set to 0,0,0.*

6.2 Constructing Points

As well as measuring points directly, they can also be constructed either by typing the point co-ordinates via the keyboard, or alternatively by constructing it at the intersection of features which have previously been measured, or constructed themselves.

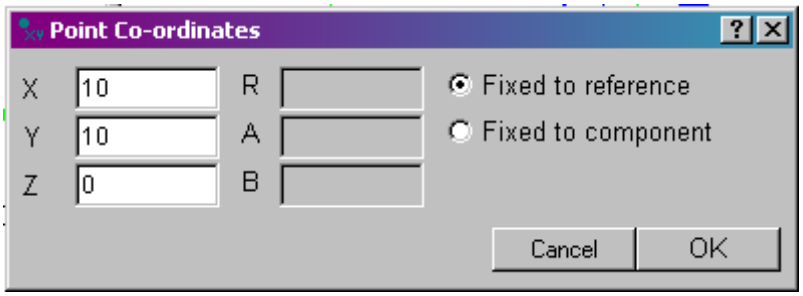
Points can be constructed at the intersection of many features, such as between 2 lines, lines and circles, or at the intersection of lines (including cylinders and cones) with planes, etc. Some of these features will produce a single intersection point, such as 2 lines, and some will have 2 or more intersection points, such as an overlapping line and circle.

One other use for points is that they can be constructed at the mid-point or end points of a line.

6.2.1 Entering the Co-ordinates for the Point

Method
To construct a point by entering the co-ordinates for the point via the keyboard, click on the Point Measure button, from the main screen. This will bring up the Point Measure window. Now click on the 'Construct' box. A prompt will now appear on the screen asking if you would like to enter the co-ordinates of the point.

If you click on yes, another window will appear, allowing you to enter the X,Y and Z (carthesian) co-ordinates or the R, A and B (Polar) values of the point with the 'R 'representing the distance from the X,Y origin, 'A' being the angle in XY, and 'B' being the elevation angle from the XY plane via the keyboard.

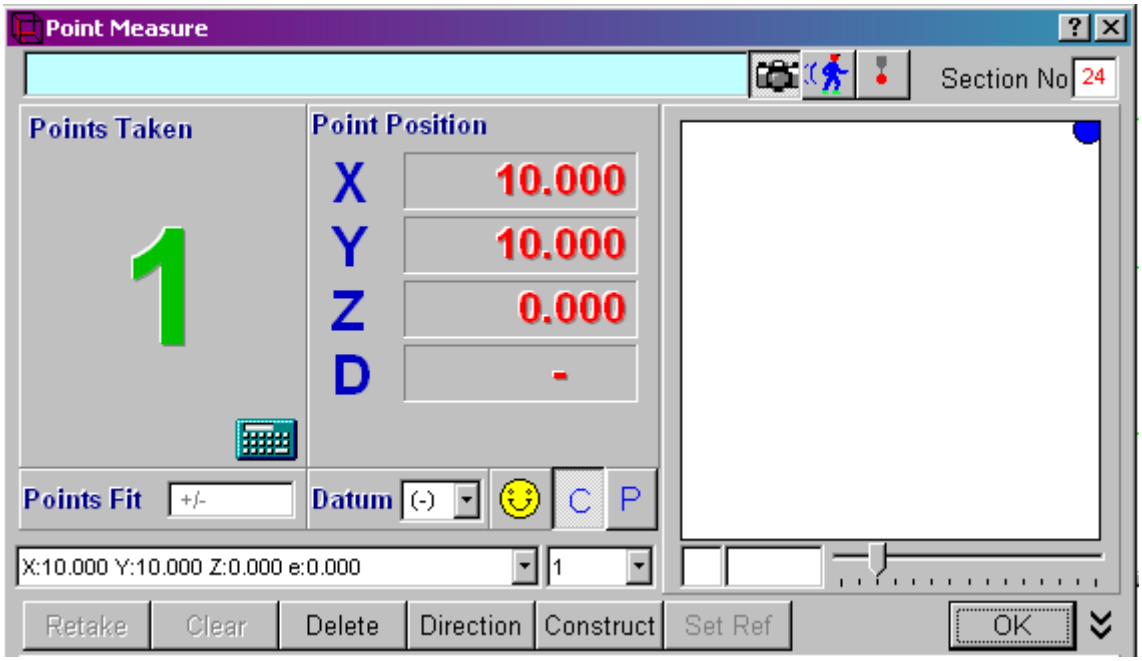


You have the choice of linking the constructed point to the reference system or to the component.

If you link the point to the component and then rotate or translate that component the point will rotate and move with that component.

If you link the point to original reference system it will say fixed to that original reference system even if you rotate or translate the component in the software.

Having entered your desired co-ordinates click 'OK' and the Point Measure window will automatically reappear. The graphics part of the window and the Points Fit box will of course be blank, but the X,Y and Z co-ordinates that you entered will be displayed in red.



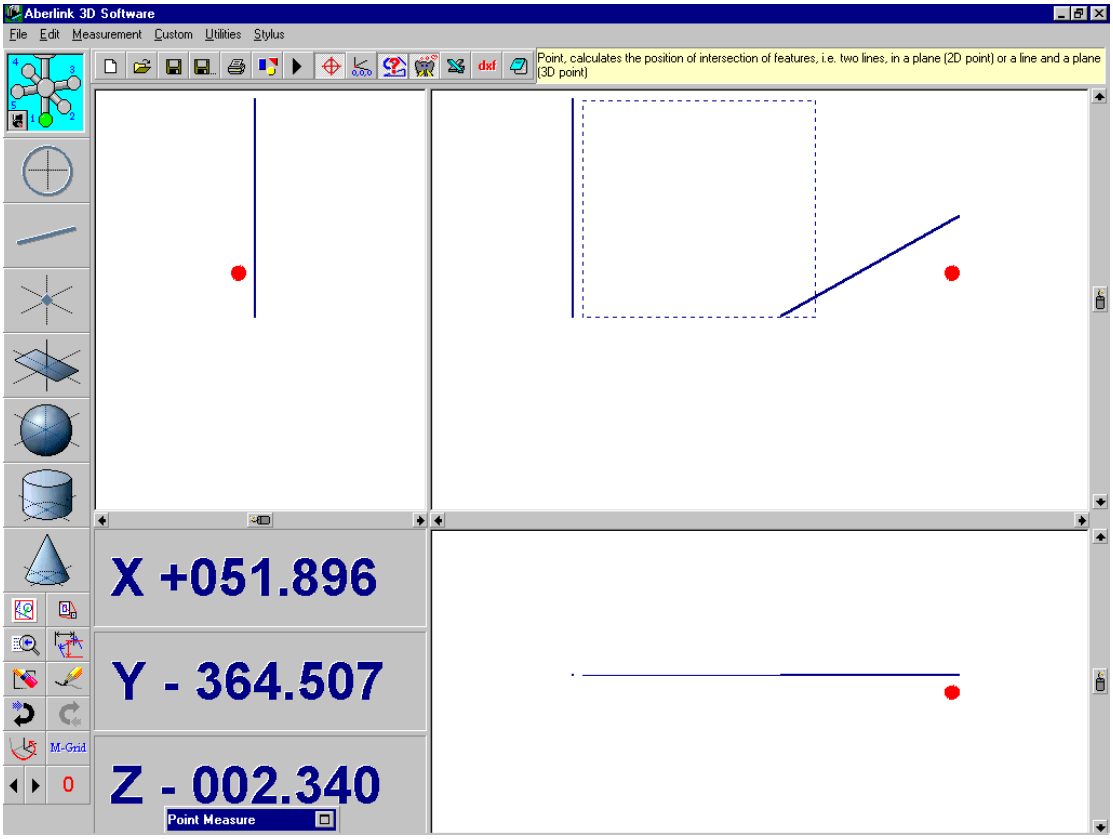
You may now click on 'OK'

The screen will now return to the Main Screen, and the point will form a part of the graphical representation.

6.2.2 Constructing points between features with a single intersection

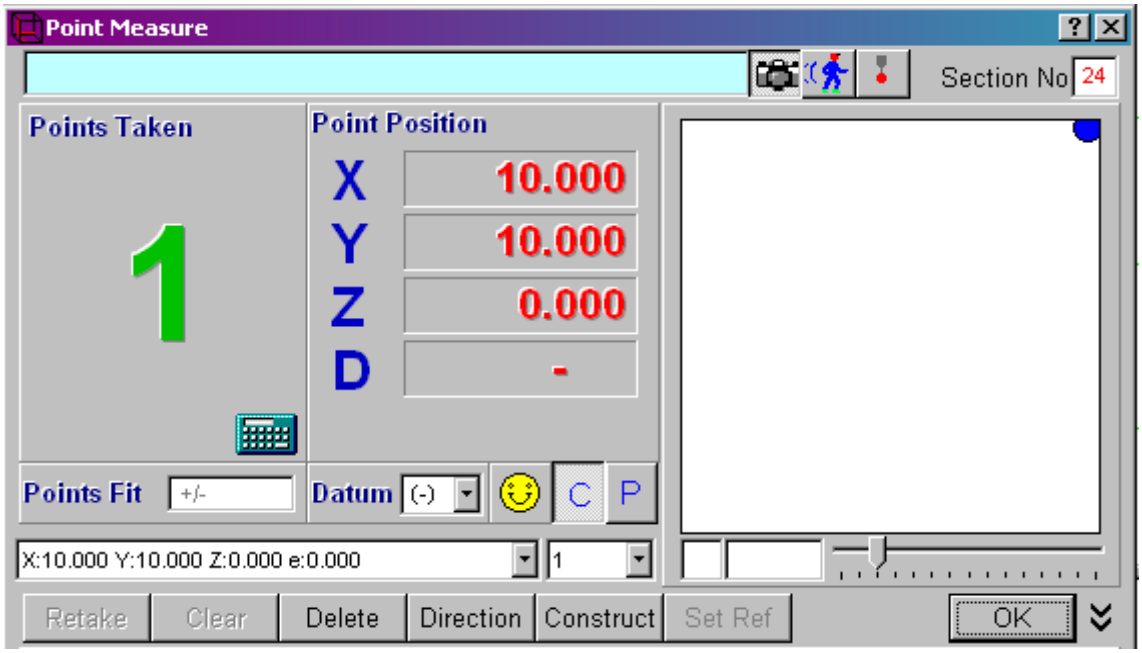
Points of intersection can be constructed not only at the intersection of features which touch, but can also be used to produce the closest point to both features, when they do not touch.

To construct such a point, click on the Point Measure button, from the main screen. This will bring up the Point Measure window. Now click on the 'Construct' box. The Point Measure window will now shrink to a small box at the bottom of the screen.

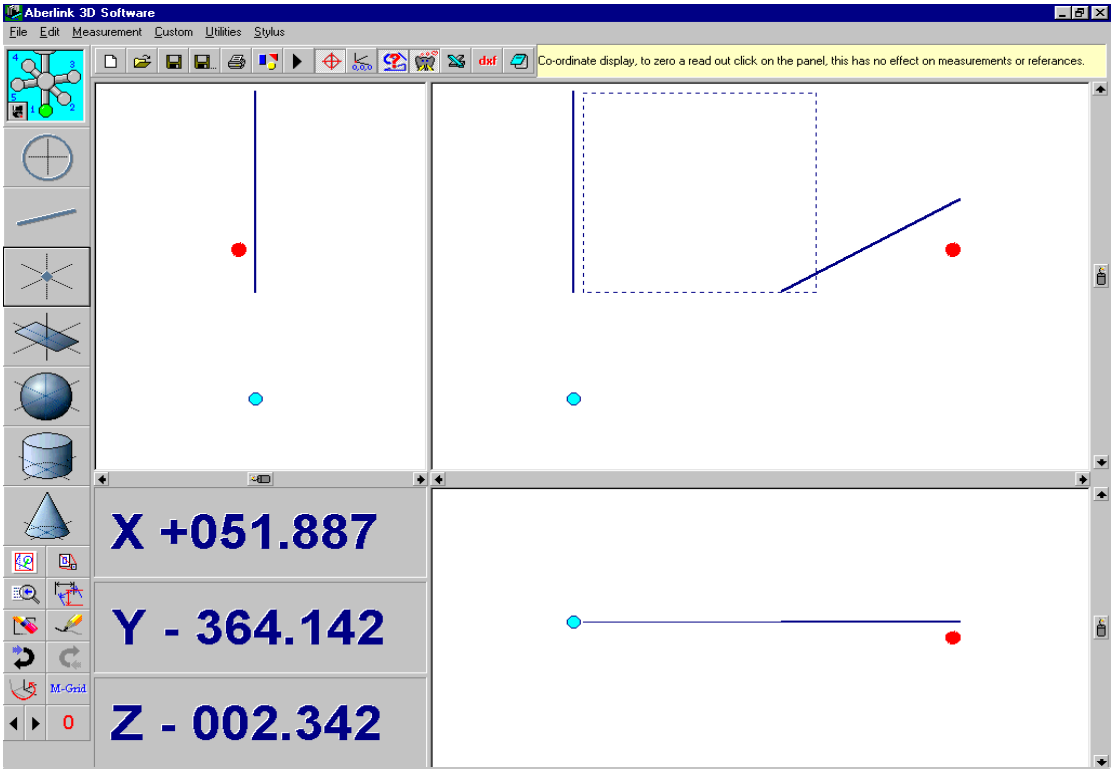

Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Point, calculates the position of intersection of features, i.e. two lines, in a plane (2D point) or a line and a plane (3D point)
X +051.896
Y - 364.507
Z - 002.340
Point Measure

Now simply select one the features, whose intersections create the desired point you want, by clicking on it.

After you have click on the second feature the Point Measure window will automatically reappear. The graphics part of the window and the Points Fit box will of course be blank, but the X,Y and Z co-ordinates of the constructed point will be displayed in red.



You may now click on ‘OK’ or ‘Set Ref’ plus ‘OK’ as appropriate, as for any point. The screen will now return to the Main Screen, and the point will form a part of the graphical representation.

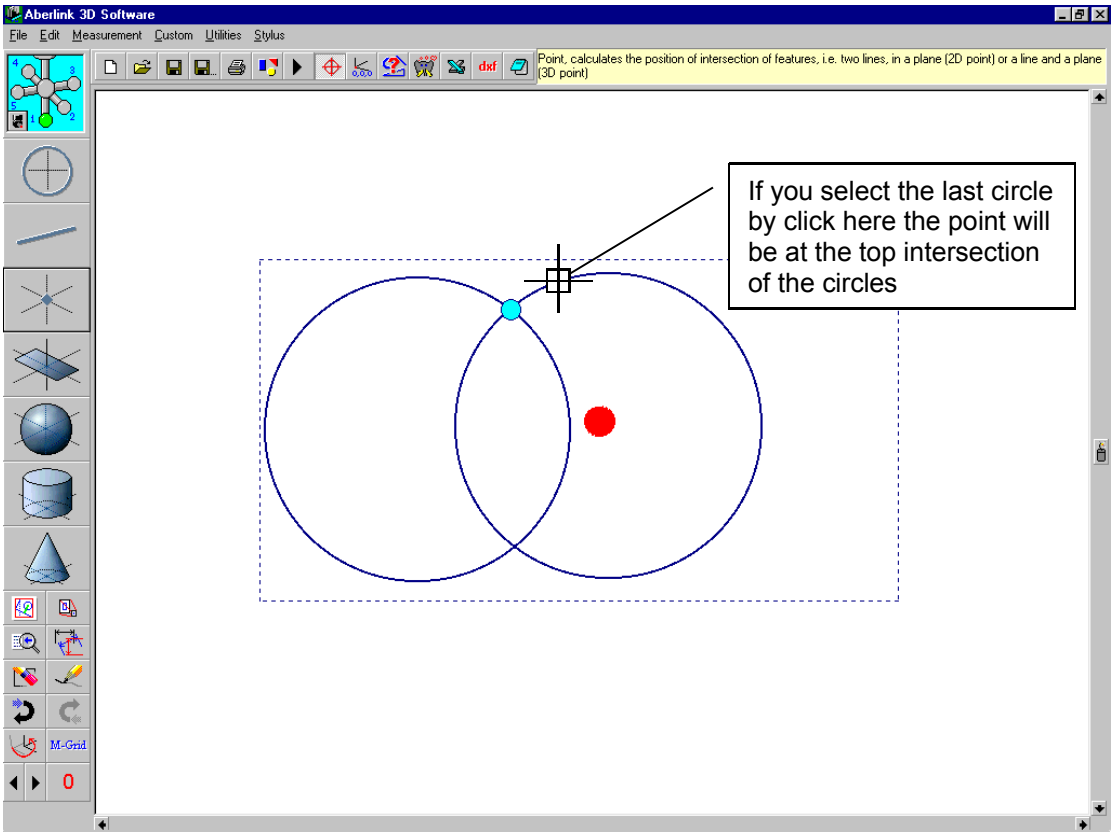

Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X +051.887
Y - 364.142
Z - 002.342

6.2.3 Constructing Points between features when there is more than one intersection

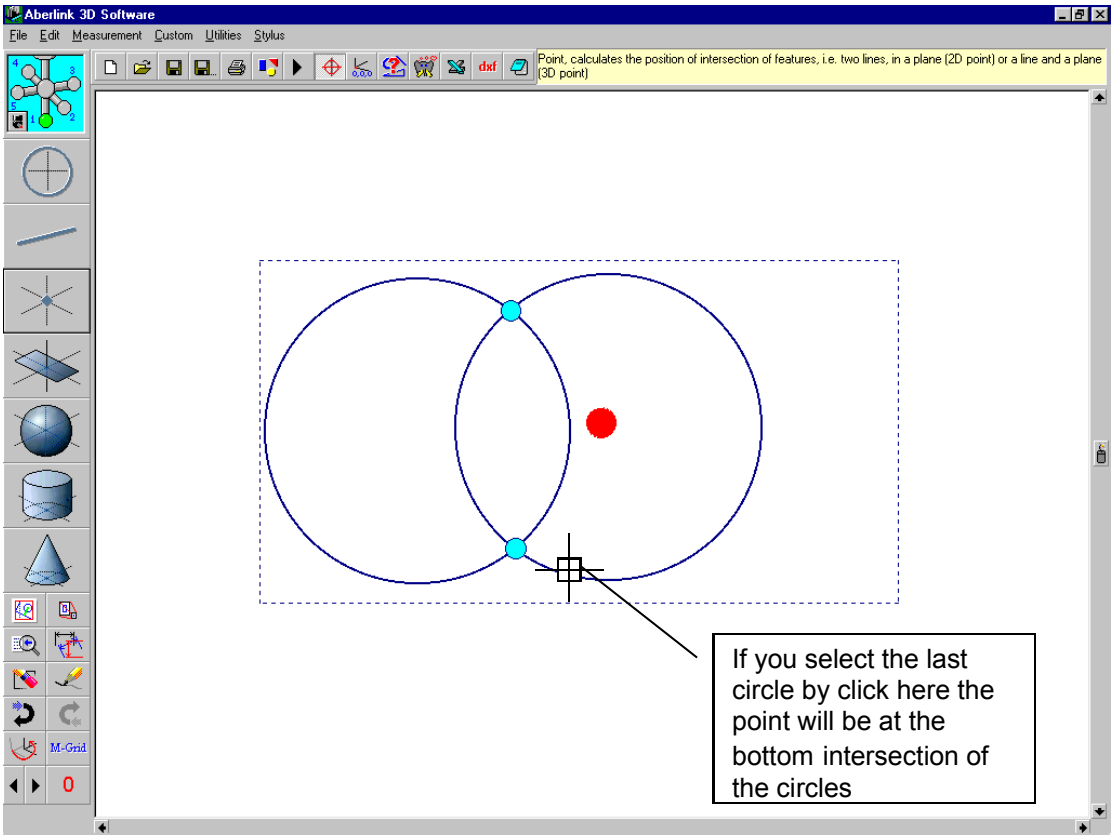
There are a few situations where the intersection of features can construct more than one point, such as two overlapping circles have 2 points of intersection. In this instance the intersection point closest to the cursor when the last feature is selected will be constructed.

> **Note** – Using the feature select function will allow you to always select a position close to the required intersection

For example when constructing an intersection point between two overlapping circles, if you select the second circle as shown, then the following intersection point will be created.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Point, calculates the position of intersection of features, i.e. two lines, in a plane (2D point) or a line and a plane (3D point)
If you select the last circle by click here the point will be at the top intersection of the circles
M-Grid
0

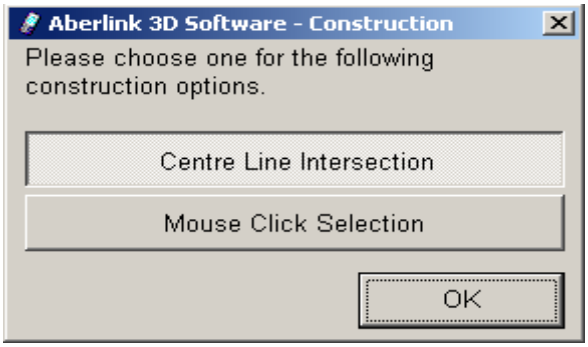
Now to select the other intersection point simply repeat the exercise, but this time select the second circle at the position shown below:


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Point, calculates the position of intersection of features, i.e. two lines, in a plane (2D point) or a line and a plane (3D point)
M-Grid
0
If you select the last circle by click here the point will be at the bottom intersection of the circles

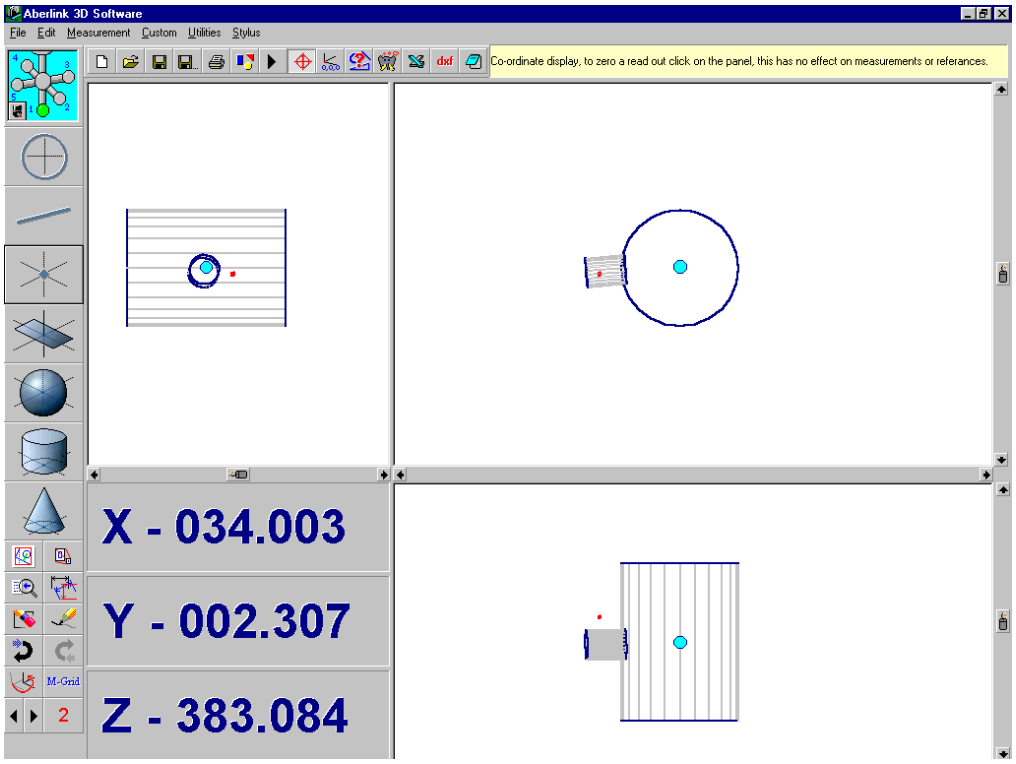
6.2.4 Constructing Points between two cylinders

A special case is the intersection of two cylinders. In this instance there are 5 possible intersection points, namely the intersection of the two centre lines, and then 2 intersections for each centre line with the edges of the other cylinder.

After you have clicked on the 2nd cylinder the above options window will appear giving you two options.

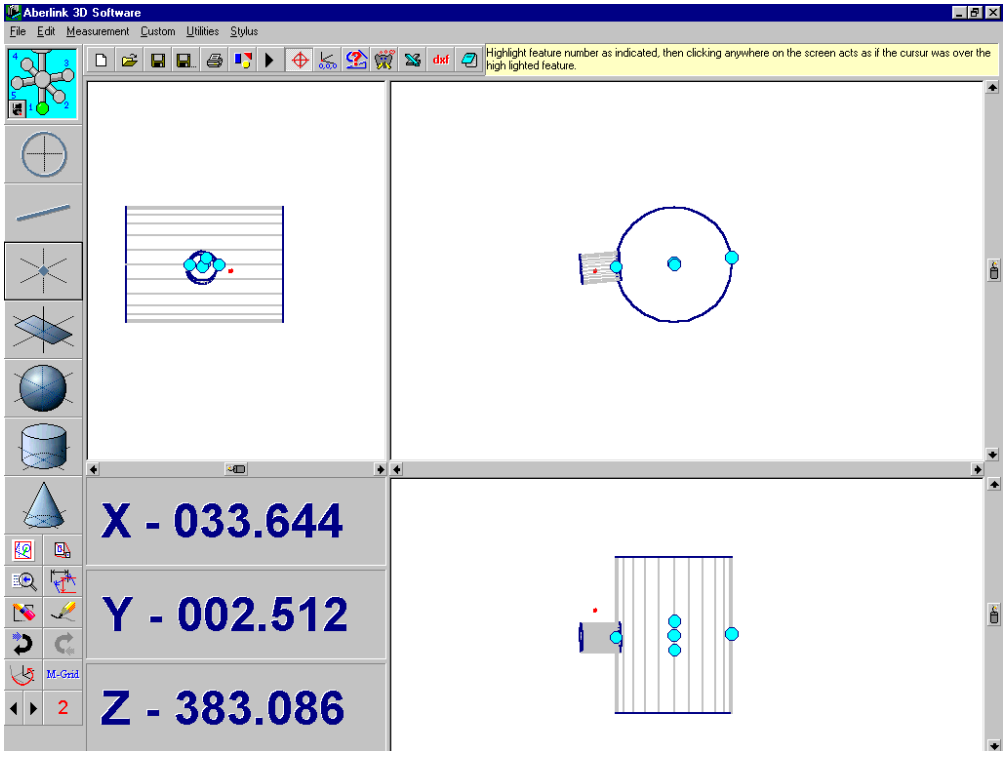

Aberlink 3D Software - Construction
Please choose one for the following construction options.
Centre Line Intersection
Mouse Click Selection
OK

If you select the first option ‘Centre Line Intersection’ the software will construct a point were the centre lines of the two cylinders meet.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 034.003
Y - 002.307
Z - 383.084
M-Grid
2

However, if you choose Mouse Click Selection, then it will construct the closest point of intersection to where you selected the second cylinder. In this way the other 4 points of intersection can be constructed as shown below:

.

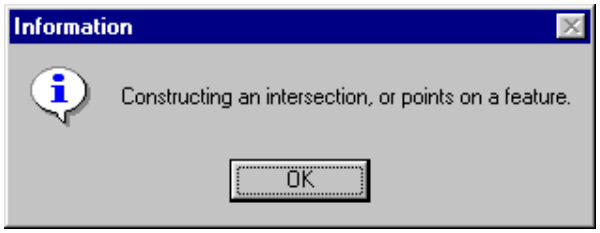

Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Highlight feature number as indicated, then clicking anywhere on the screen acts as if the cursur was over the high lighted feature.
X - 033.644
Y - 002.512
Z - 383.086
M-Grid
2

6.2.5 Constructing a Point at the Mid-Point or the end of a Line

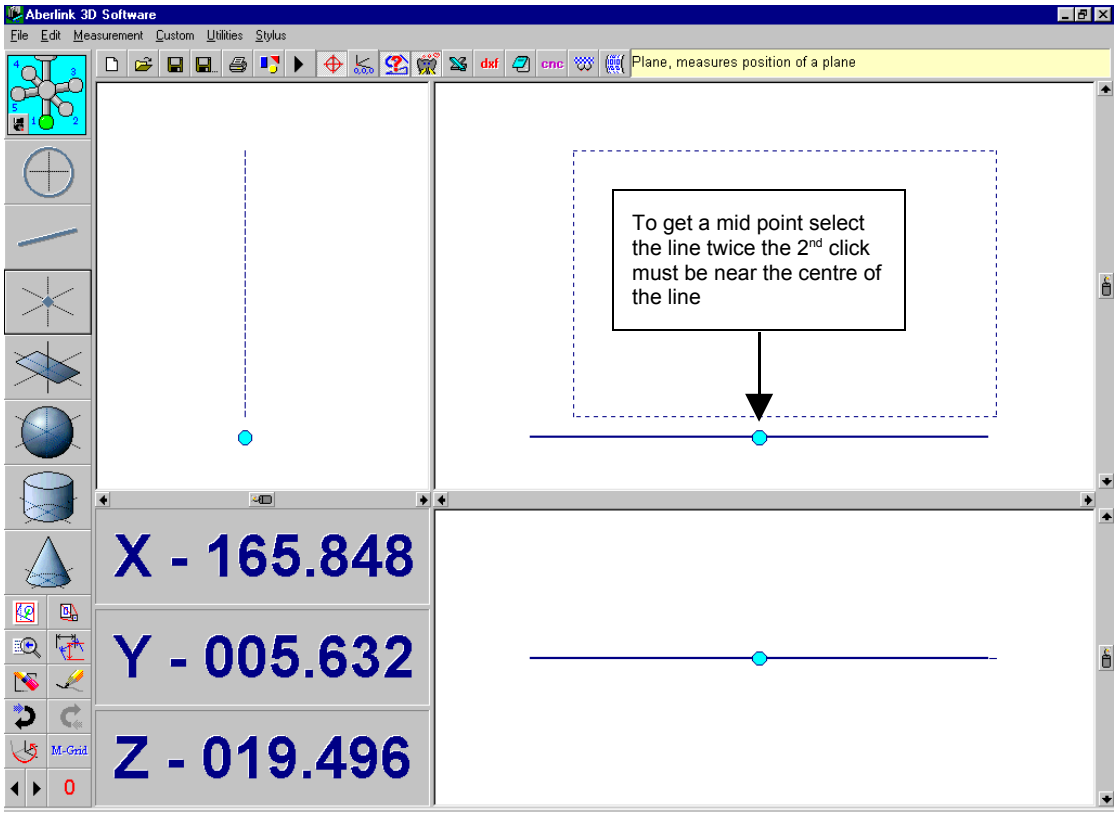
One further useful technique allows the construction of a point at the mid-point or the end of a line.

To construct such a point, click on the Point Measure button, from the main screen. This will bring up the Point Measure window. Now click on the 'Construct' box. and the Point Measure window will now shrink to a small box at the bottom of the screen.

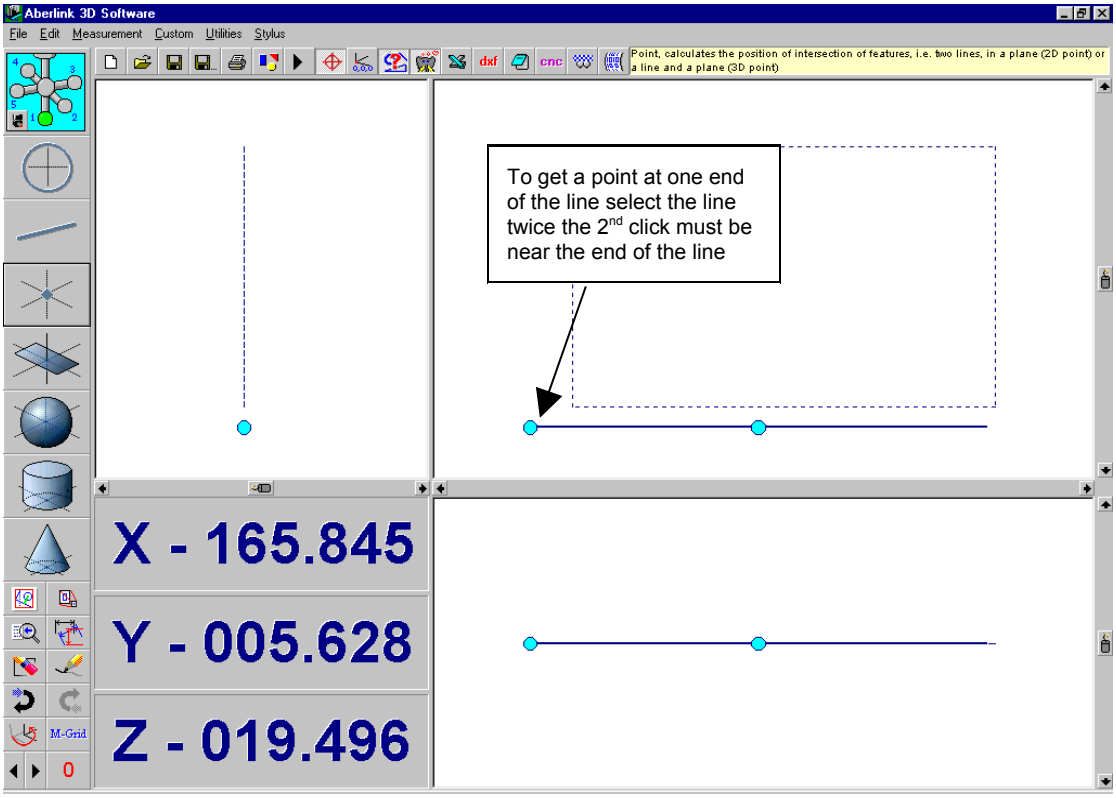
Now simply select the line twice. After you have selected it for the first time, a prompt will appear, to confirm what is happening:



Click OK, and then click on the line a second time near it's centre. Now the Point Measure window will automatically reappear. The graphics part of the window and the Points Fit box will of course be blank, but the X,Y and Z co-ordinates of the constructed point will be displayed in red. You may now click on 'OK'. The screen will now return to the Main Screen, and a point will have been constructed at the mid-point of the line.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Plane, measures position of a plane
To get a mid point select the line twice the 2nd click must be near the centre of the line
X - 165.848
Y - 005.632
Z - 019.496

To construct the end point of a line repeat exactly the same process but this time, the 2nd click must be near to the end of the line at which you wish to construct the point:


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Point, calculates the position of intersection of features, i.e. two lines, in a plane (2D point) or a line and a plane (3D point)
To get a point at one end of the line select the line twice the 2nd click must be near the end of the line
X - 165.845
Y - 005.628
Z - 019.496
M-Grid
0

7.0 THE SPHERE UNIT

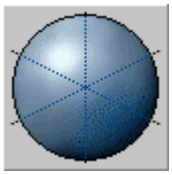
Spheres can be measured directly by taking 4 or more measurement points on the surface of the sphere.

7.1 Measuring Spheres directly

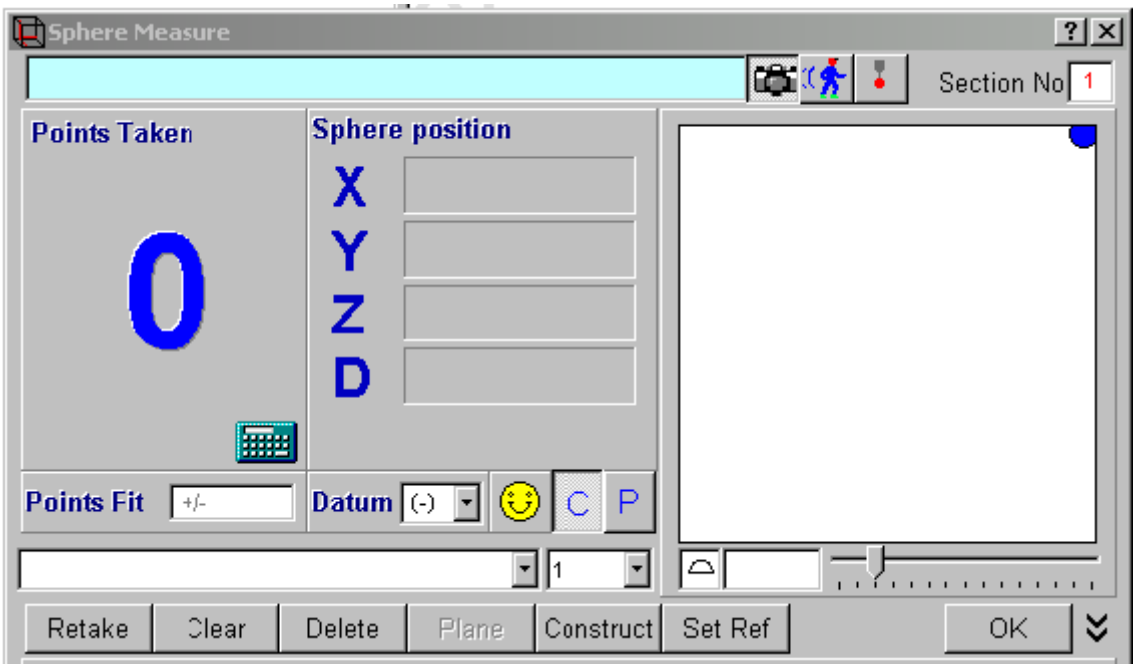
Background
The software will construct the 'best fit' sphere through the measurement points taken.

In order to determine whether it is an internal or external sphere, and apply the correct stylus radius offset, the software will look at the direction of motion of the probe when taking the measurement points.
As a sphere is a 3 dimensional feature, no reference planes will need to be defined.

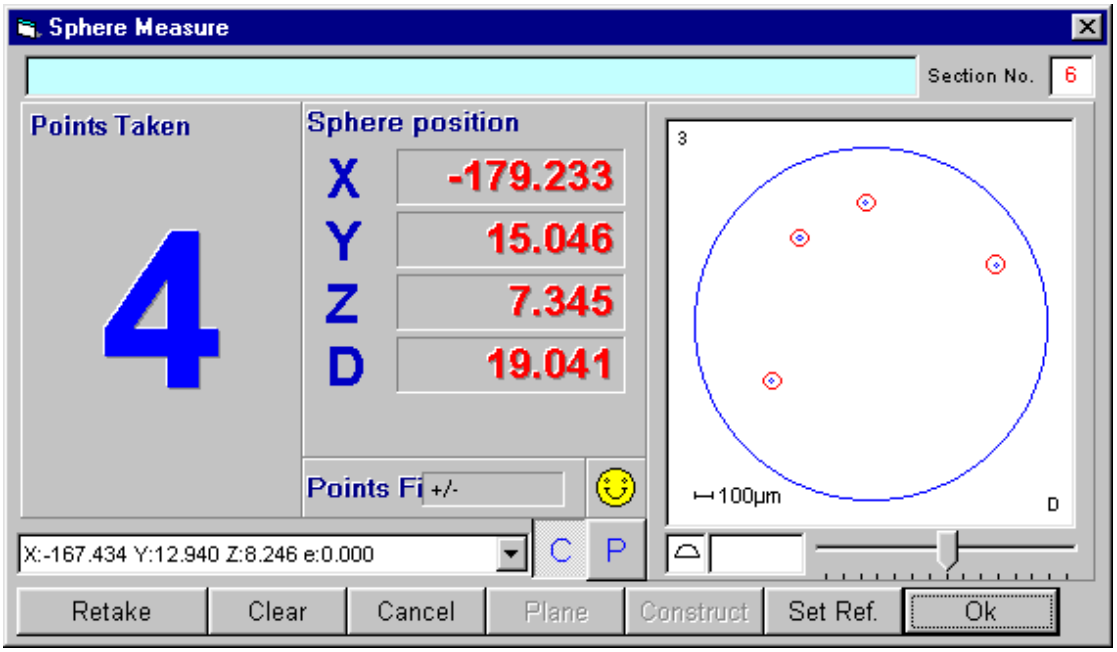
Method
To measure a sphere, click on the Sphere Measure button, from the main screen:



This will bring up the Sphere Measure window:



Now take measurement points on the sphere that you wish to measure. After 4 points a pictorial representation of the sphere will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The X,Y and Z values displayed represent the centre of the sphere and the D value represents its diameter.

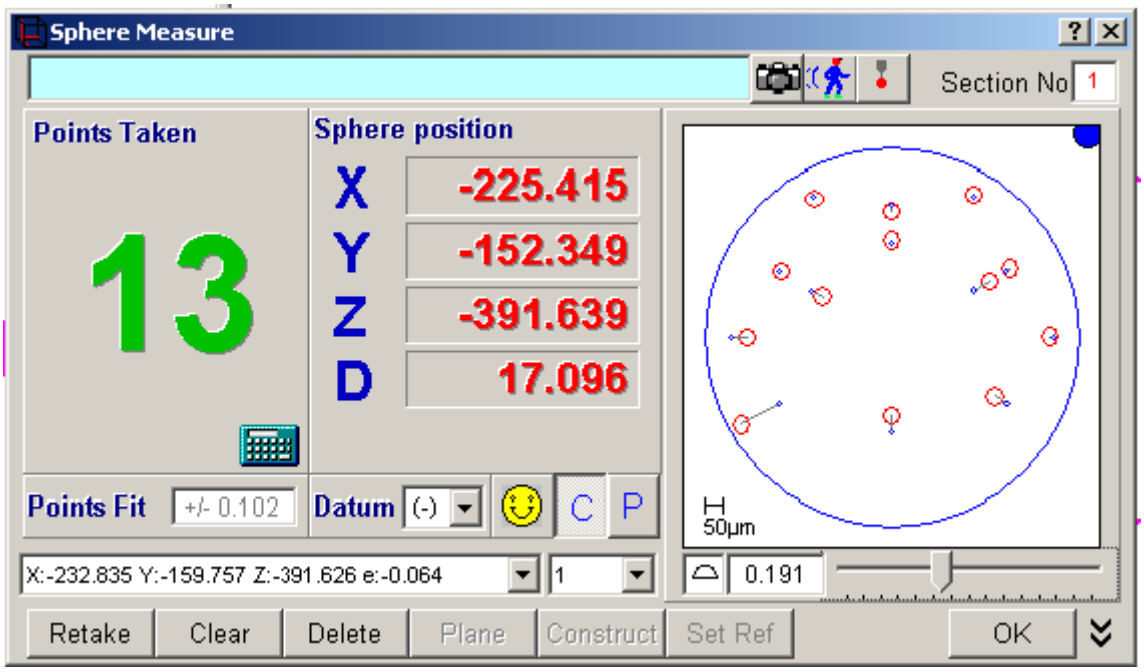


A sphere can always be calculated to fit through four points, therefore for the first 4 points the profile value for the sphere, shown under the graphical representation, will remain blank.

It is therefore good practice to take at least one more point. On the fifth point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated 'best fit' sphere and a value for the profile of this sphere.

In general the more points taken on a sphere, the more information can be gleaned, and a minimum of 5 points is recommended. Similarly the points taken should cover as much of the surface of the sphere as possible.

The graphical representation of the sphere shows the points taken as a red circle with a blue line representing the best fit sphere through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.



If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points using the retake or clear buttons.

7.1.1 When to OK a Sphere, and when to reference it.

If you select ‘OK’ the Sphere Measure window will close and the centre co-ordinates of the sphere will be relative to any references previously set, (or if none have been set then to the machine’s own reference point).

If however, you click Set Ref before OK, then the centre of the sphere will be set to 0,0,0.

> ***Note*** *– Only one sphere can ever be set as a reference at any point in time. If a second sphere (or other point) is set as a reference, this will override the initial reference point, and the centre co-ordinates of the second sphere or point will be set to 0,0,0.*

8.0 THE CYLINDER UNIT

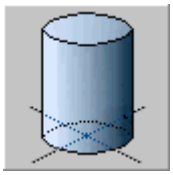
The cylinder unit can be used to measure cylindrical shapes (holes or shafts etc.) by taking 5 or more measurement points on their surface.

8.1 Measuring Cylinders directly

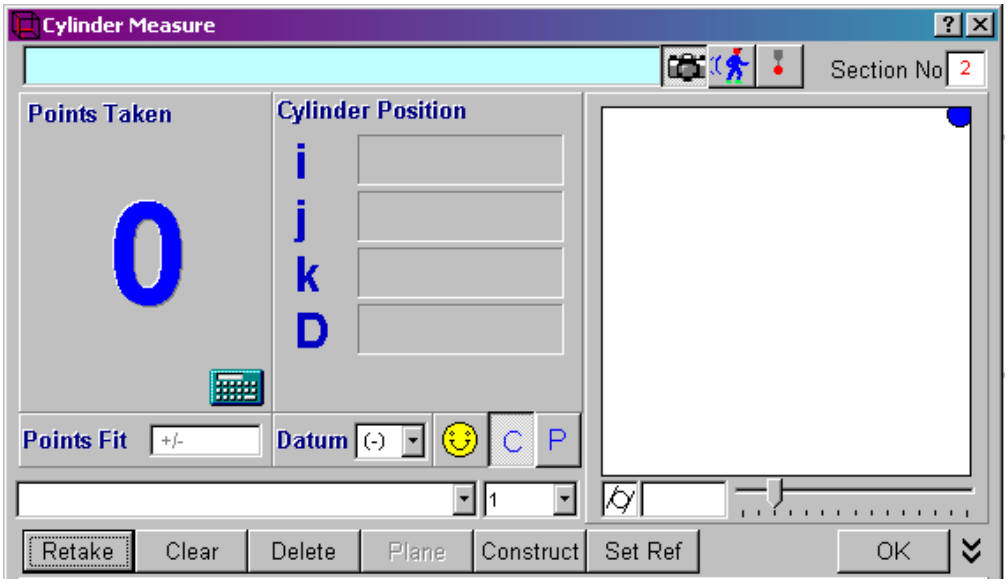
Background
The software will construct the 'best fit' cylinder through the measurement points taken. In order to determine whether it is an internal cylinder (hole) or external cylinder (shaft etc.), and apply the correct stylus radius offset, the software will look at the direction of motion of the probe when taking the measurement points.

As a cylinder is a 3 dimensional feature, no reference planes will need to be defined.

Method
To measure a cylinder, click on the Cylinder Measure button, from the main screen:

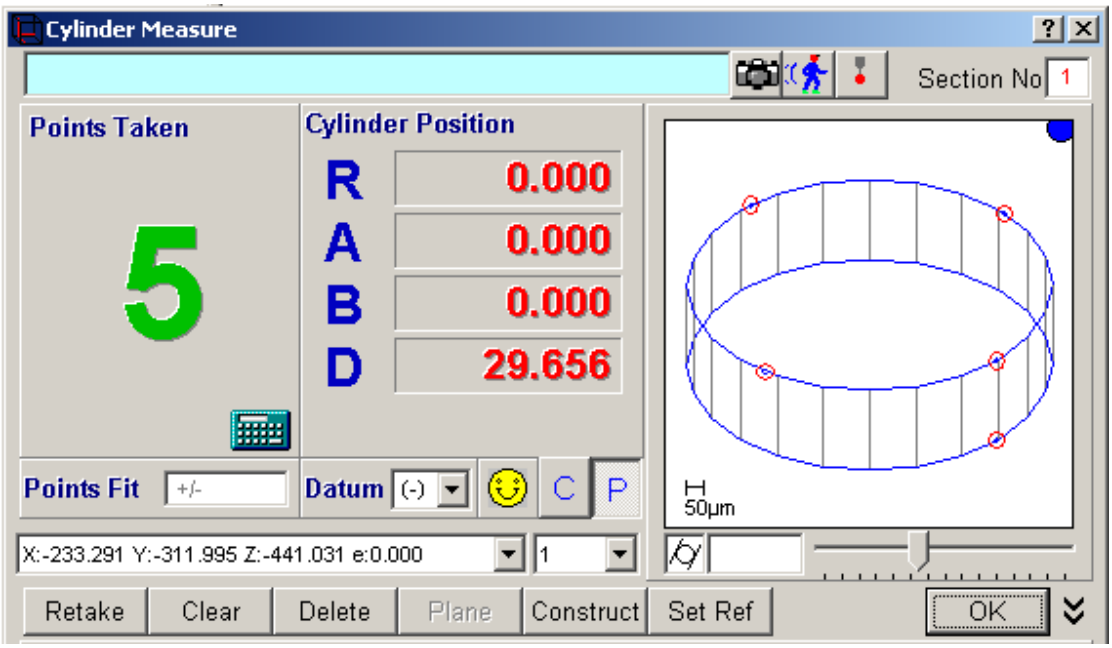


This will bring up the Cylinder Measure window:



Now take measurement points on the cylinder that you wish to measure. After 5 points a pictorial representation of the cylinder will appear in the graphics

part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the ‘Points Taken’ box. The I,J and K values displayed represent direction vector for the cylinder, and the D value represents its diameter.

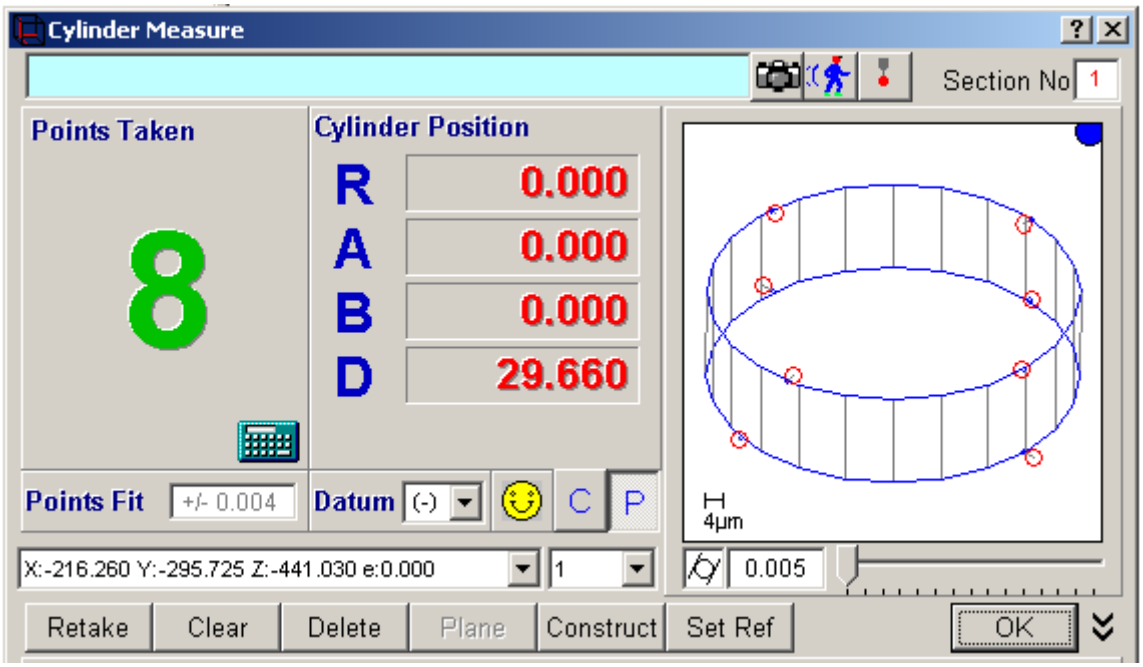


The software requires a minimum of five points in order to calculate a cylinder, therefore for the first 5 points the cylindricity value for the cylinder, shown under the graphical representation, will remain blank.

It is therefore good practice to take at least one more point. On the sixth point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated ‘best fit’ cylinder and a value for it’s cylindricity.

In general the more points taken on a cylinder, the more information can be gleaned, and a minimum of 6 points is recommended. Similarly the points taken should cover as much of the surface of the cylinder as possible.

The graphical representation of the cylinder shows the points taken as a red circle with two blue circles joined by grey lines representing the best fit cylinder through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.



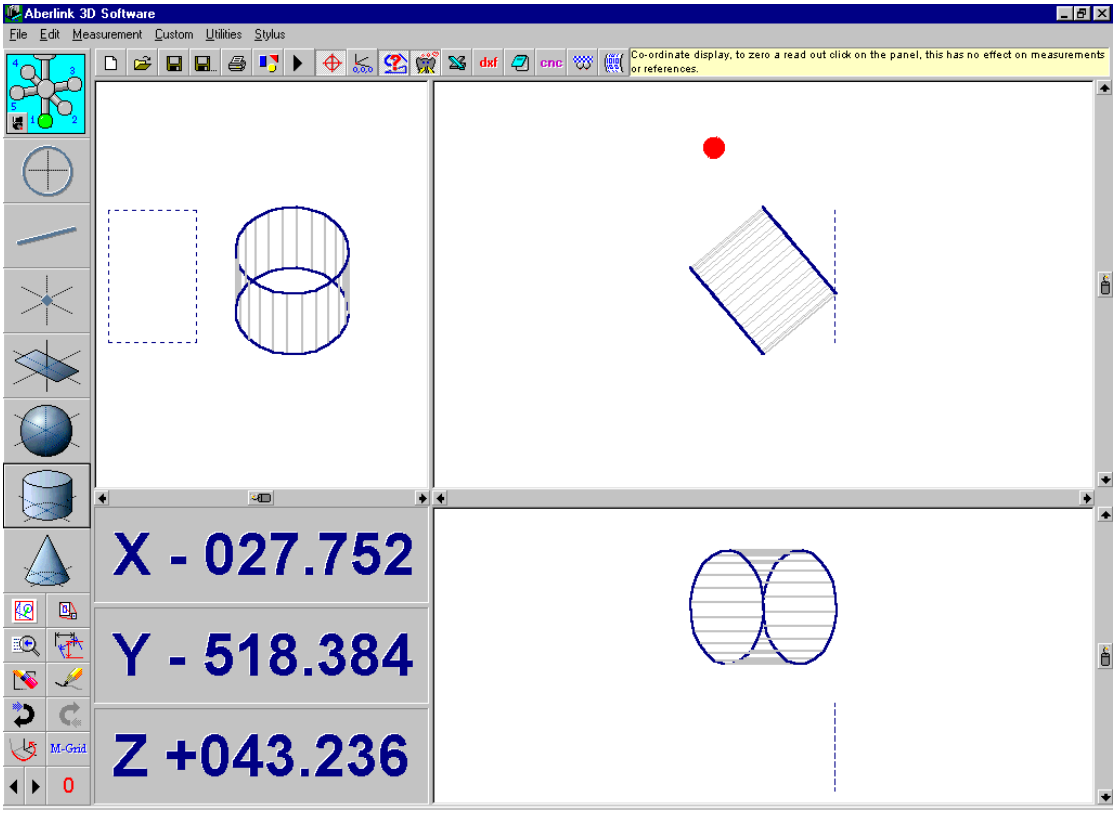
If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points using the retake and clear buttons.

8.1.1 When to OK a Cylinder, and when to reference it.

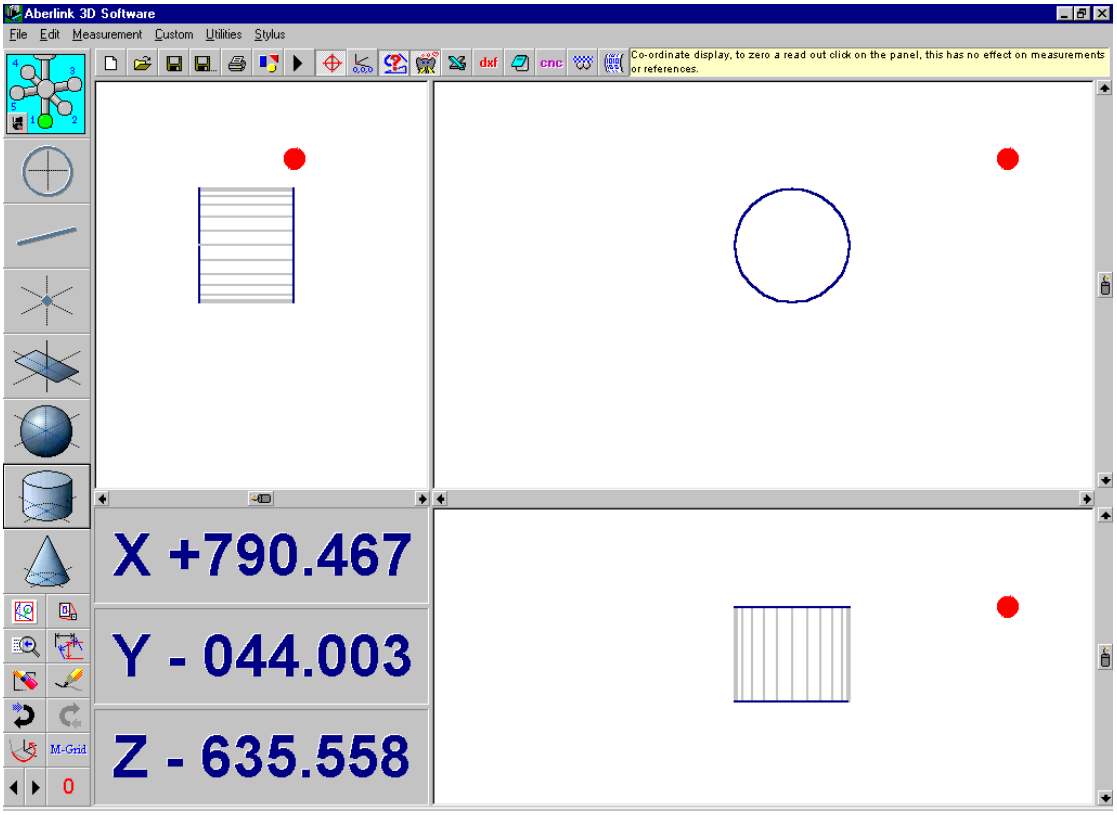
If you OK the cylinder, the measurement window will close, and the program will return to the main screen.

The graphical representation of the cylinder should be shown as two solid blue circles representing the ends of the cylinder, with grey lines joining them. This should be visible in all of the 3 views

The angle at which the cylinder is shown will be relative to any previous references that have been set, or relative to the machine axes if no references have been set.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or references.
X - 027.752
Y - 518.384
Z +043.236
M-Grid
0

However, if you set the cylinder as a reference by clicking on the 'Set Ref' button before OK, then the axis of the cylinder will be aligned in the Main Screen:


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or references.
X +790.467
Y - 044.003
Z - 635.558
M-Grid
0

If a second cylinder is then set as a reference, this cylinder will supersede the first as the primary reference, but the original cylinder will still be used to define any degrees of freedom that are not defined by the second cylinder.

Similarly a cylinder can be used as a secondary alignment with a Plane. Please note that in the software it is the reference that is set last which becomes the primary alignment.

9.0 THE CONE UNIT

The cone unit can be used to measure conical features such as countersinks and circular chamfers.

9.1 Measuring Cones directly

Background
The software will construct the 'best fit' cone through the measurement points taken.

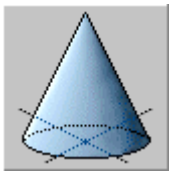
If the user is not able to define the axis of the cone before starting the measurement, then the measurement points must be taken in a specific order. A first set of measurement points must be taken near one end of the cone, followed by a second set at the opposite end of the cone.

If the user is able to define the axis of the cone, then the measurement points may be taken in any order, and a minimum of 6 points is required.

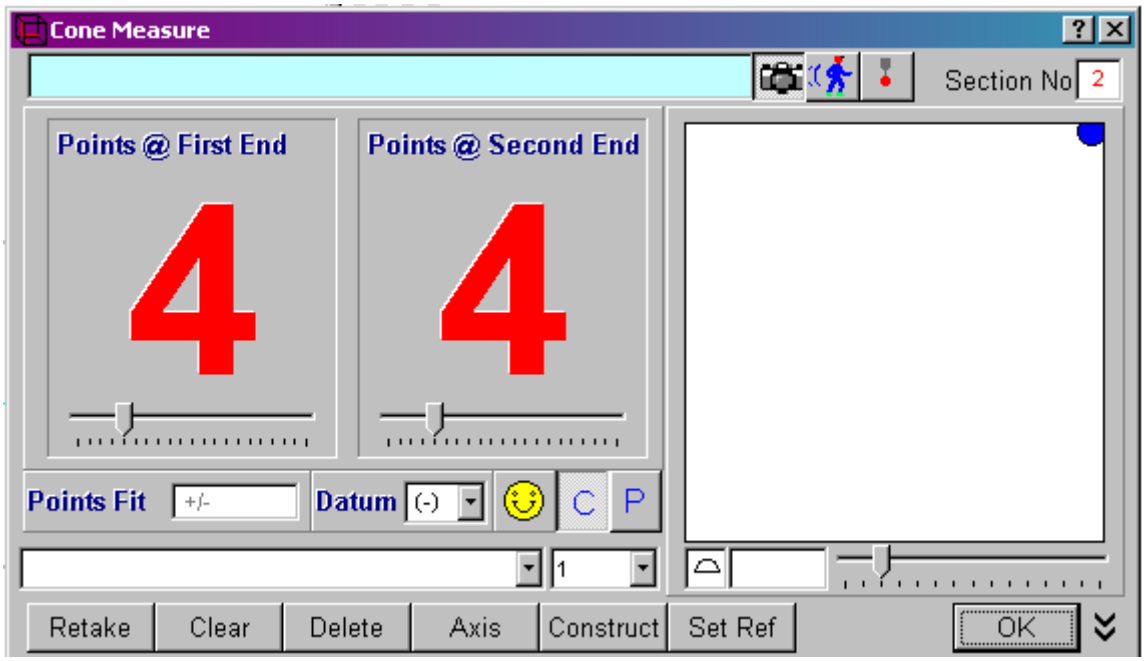
In order to determine whether it is an internal or external cone, and apply the correct stylus radius offset, the software will look at the direction of motion of the probe when taking the measurement points.

9.1.1 Measuring cones without defining their axis

Method
Click on the Cone Measure button, from the main screen:



This will bring up the Cone Measure window:

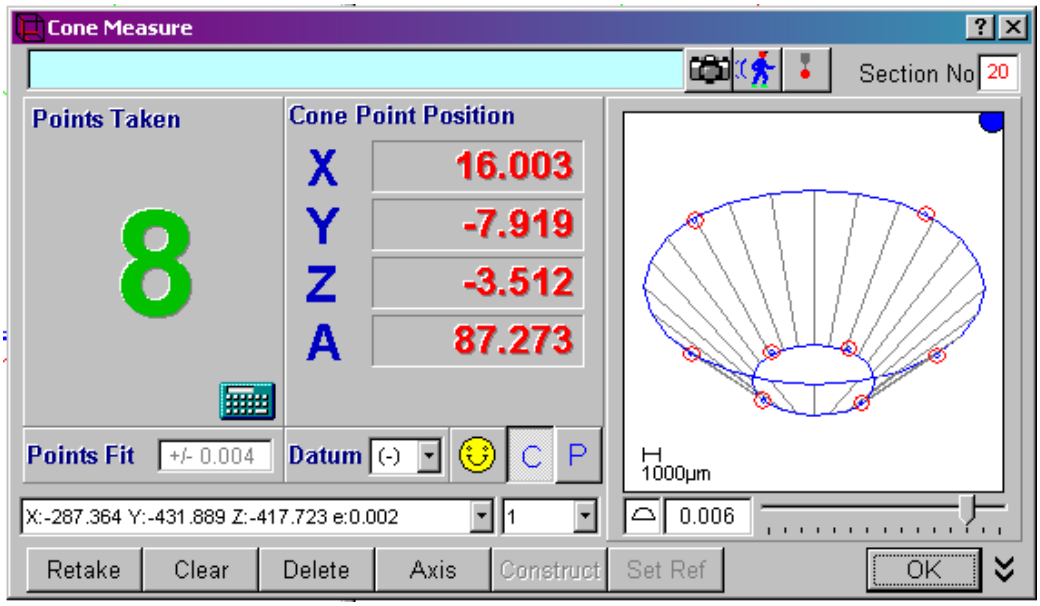


Note that instead of having one large figure on the screen displaying points taken, there are now two figures on the screen, one under the heading ‘Points at First End’ and one under the heading ‘Points at Second End’. These numbers can be adjusted using the slider bar in each box.

The split box denotes that it is important to take some measurement points (minimum 3) near one end of the cone, and some points (again minimum 3) at the other end

Each time you take a measurement point at the first depth in the cone, the number under ‘Points at First End’ will reduce by 1, until it is zero. You must then start to take points at the other depth on the cone. Now for each point taken the number under ‘Points at Second End’ will reduce by 1.

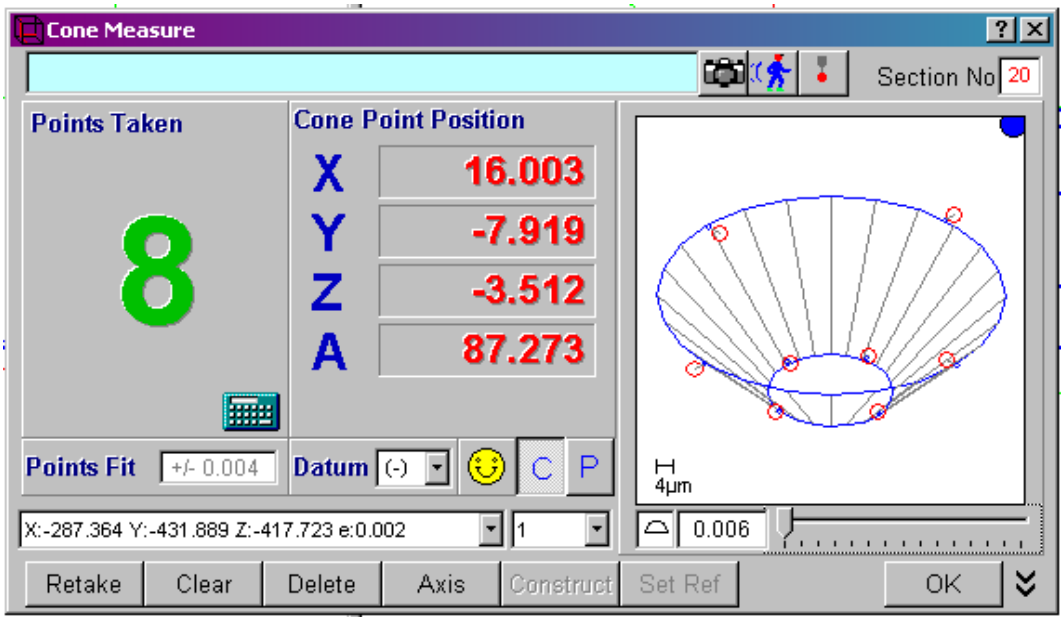
When you have reduced both numbers to zero the window will revert to the usual single figure of ‘Points Taken’ and a pictorial representation of the cone will appear on the graphics part of the window. The X,Y and Z values displayed represent the co-ordinate for the apex of the cone, and the A value represents its included angle.



You may now continue to take extra measurement points at any depth on the cone. After each point the software will update the graphical representation and calculate fresh values for the fit of the points on the calculated ‘best fit’ cone, and for the profile of the cone.

In general the more points taken on a cone, the more information can be gleaned, and a minimum of 6 points is required. Similarly the points taken should cover as much of the surface of the cone as possible.

The graphical representation of the cone shows the points taken as a red circle with a blue circle and a point joined by grey lines representing the best fit cone through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

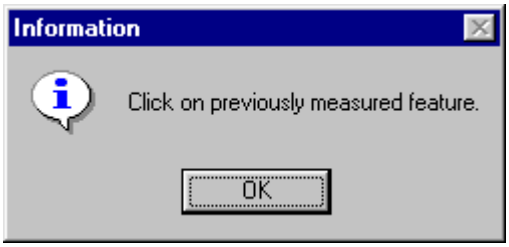


If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points using the retake or clear buttons.

12.1.2 Measuring cones having defined their axis

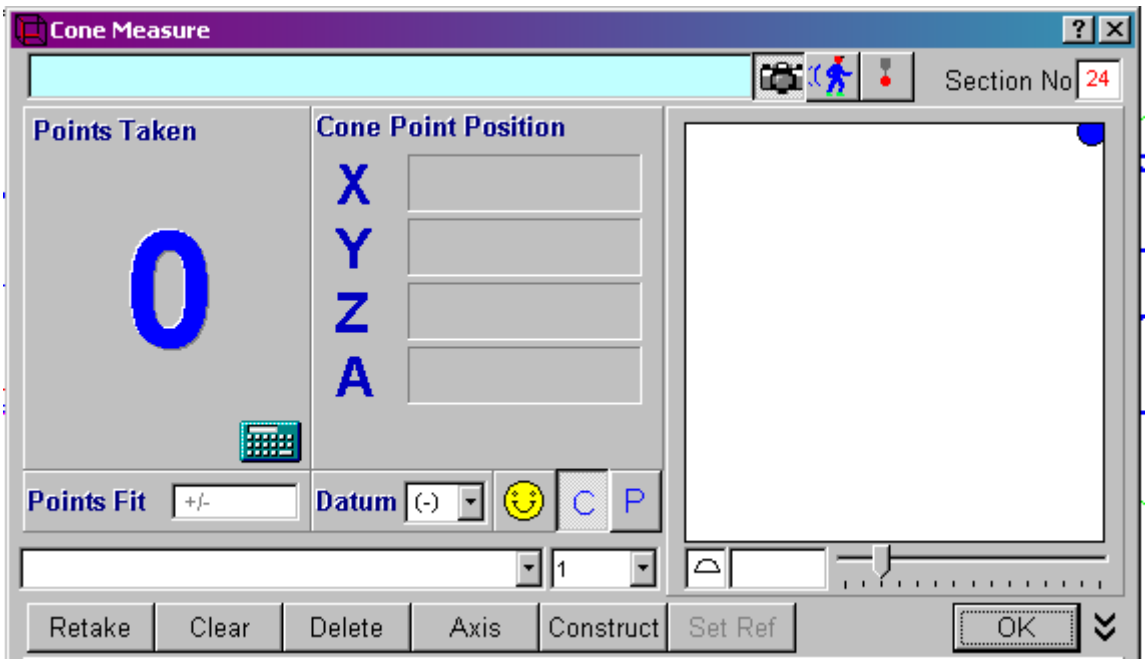
Method

Click on the Cone Measure button, from the main screen. This will bring up the Cone Measure window. Now click on the 'Axis' button, and the following window will appear:

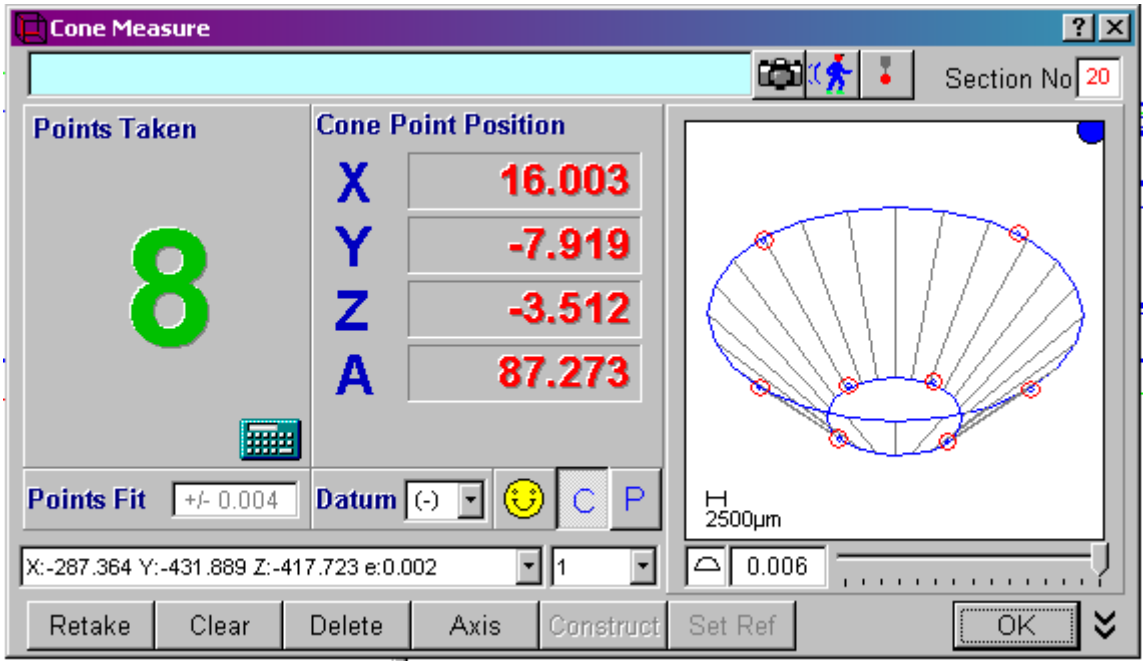


You can now define the direction of the axis of the cone, Click the 'OK' button in the above prompt window and then click on a previously measured feature which defines the axis of the cone.

The measurement window will revert to the usual Zero figure for 'Points Taken', and you can now proceed to take measurement points on the cone in any order.



After 6 points a pictorial representation of the cone will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the ‘Points Taken’ box. The X,Y and Z values displayed represent the co-ordinate for the apex of the cone, and the A value represents its included angle.



You may now continue to take extra measurement points at any depth on the cone. After each point the software will update the graphical representation and calculate fresh values for the fit of the points on the calculated ‘best fit’ cone, and for the profile of the cone.

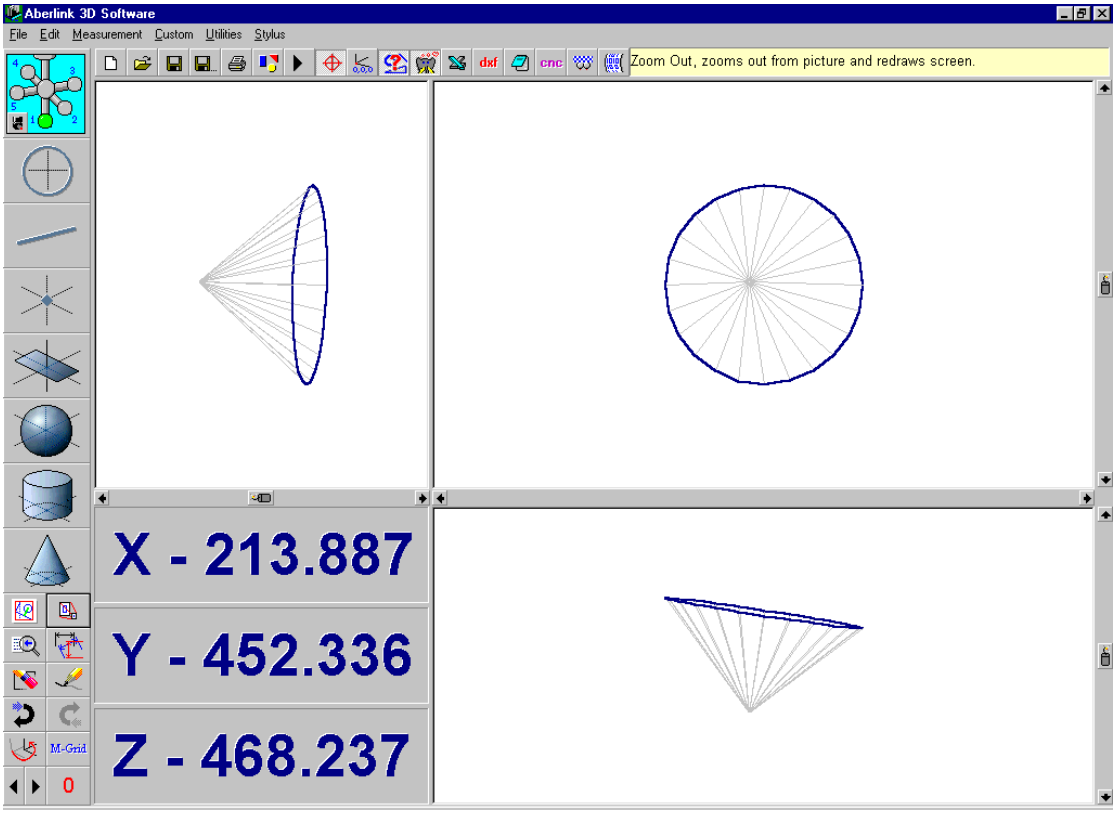
In general the more points taken on a cone, the more information can be gleaned, and a minimum of 6 points is required. Similarly the points taken should cover as much of the surface of the cone as possible.

9.1.3 When to OK a Cone, and when to reference it.

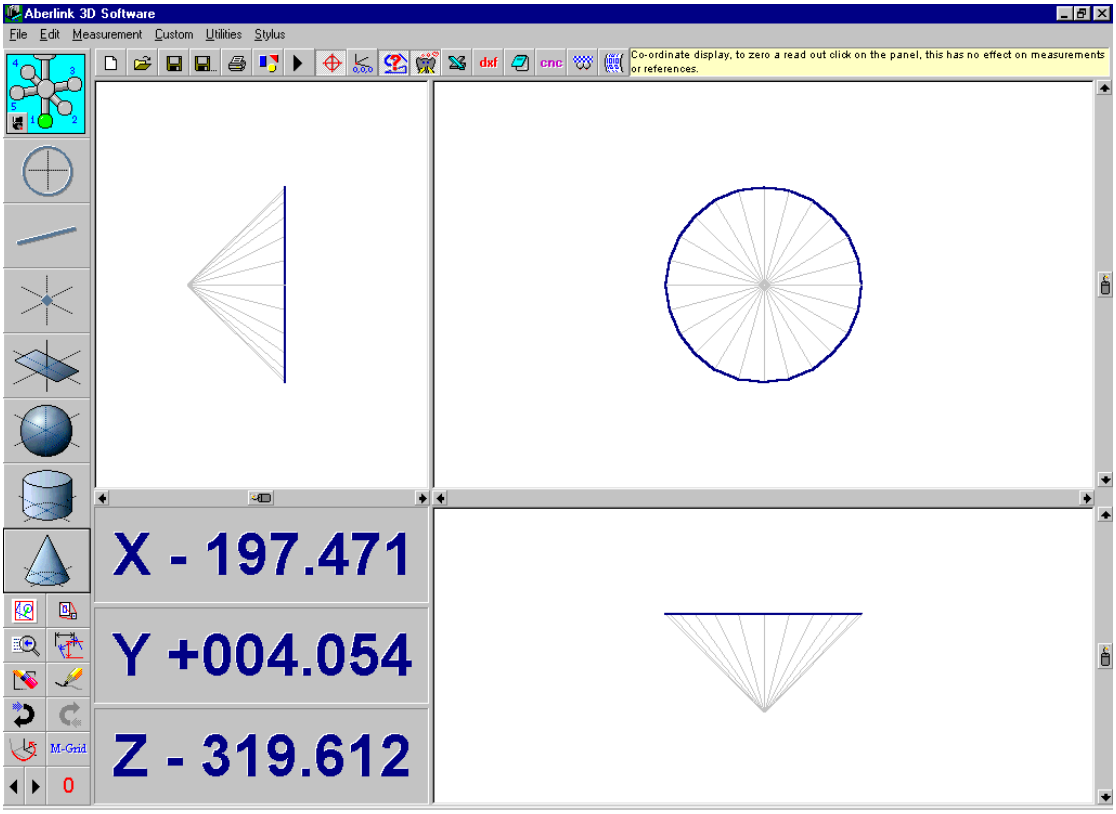
If you OK the cone, the measurement window will close, and the program will return to the main screen.

The graphical representation of the cone will be shown as a solid blue circle representing the end of the cone, with grey lines joining it to the apex. This should be visible in all of the 3 views

The angle at which the cone is shown will be relative to any previous references that have been set, or relative to the machine axes if no references have been set:



However, if you set the cylinder as a reference by clicking on the 'Set Ref' button before OK, then the axis of the cylinder will be aligned in the Main Screen:


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or references.
X - 197.471
Y +004.054
Z - 319.612
M-Grid
0

Note – Only one cone can ever be set as a reference at any point in time. If a second cone (or line or point) is set as a reference, this will override the initial reference, and the graphical representation will now re-align itself to the second cone.

If a second cone is then set as a reference, this cone supersedes the first as the primary reference, but the original cone will still be used to define any degrees of freedom that are not defined by the second cone.

Similarly a cone can be used as a secondary alignment with a Plane. Please note that in the software it is the reference that is set last which becomes the primary alignment.

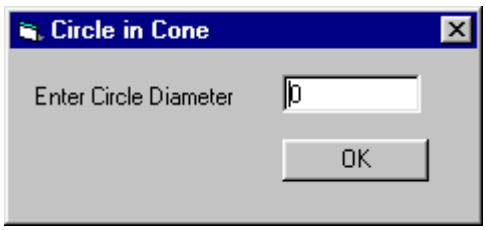
12.2 Measuring the maximum diameter of a cone.

Often it will be important to know the diameter of the cone at its large end, where it meets a plane. Obviously the measured cone will only have information over the area that the actual measurement points were taken.

In this case the required diameter can be obtained by constructing a circle at the intersection of the cone and the plane.

12.3 Determining the Position on a cone of a given diameter.

Sometimes it may be necessary to determine at what position on a cone is a given diameter. This is done by constructing a circle and selecting the cone twice. This will bring up the following prompt:

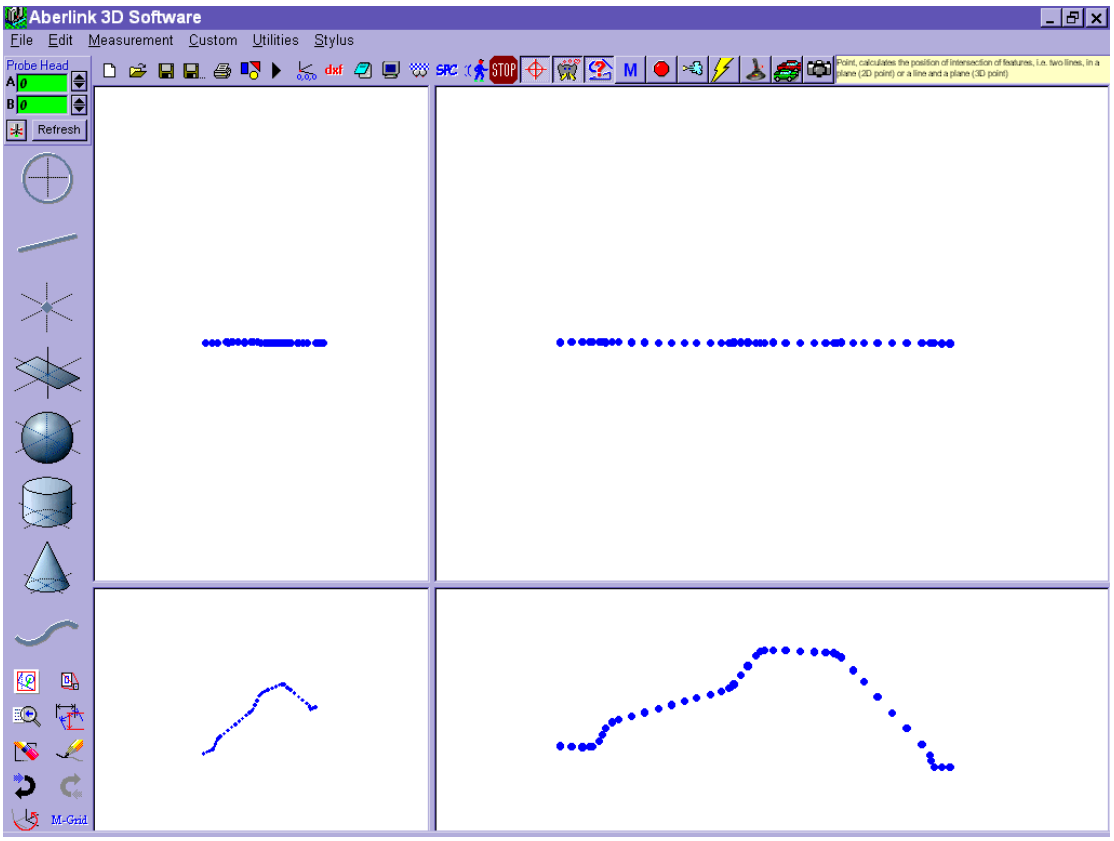


Now simply enter the desired diameter via the keyboard, and then click ‘OK’, and a circle will be constructed on the cone at this diameter.

12.0 THE CURVE MEASUREMENT UNIT

The Curve Measure function allows the user to take multiple points over a surface or surfaces without fitting a geometric shape through the points. In effect this is simply a method for gathering point cloud data.

When using this function points can be taken on the surface of a component that possibly form a complex shape. Because the software will not know what direction to compensate for the radius of the stylus these points will represent the centre of the stylus ball:



This information is useful when exported to a CAD package as a dxf file.

Similarly it is possible to import the dxf file for the shape that has been measured and perform a best fit of the measured data to the dxf file. Please see the chapter regarding dxf file.

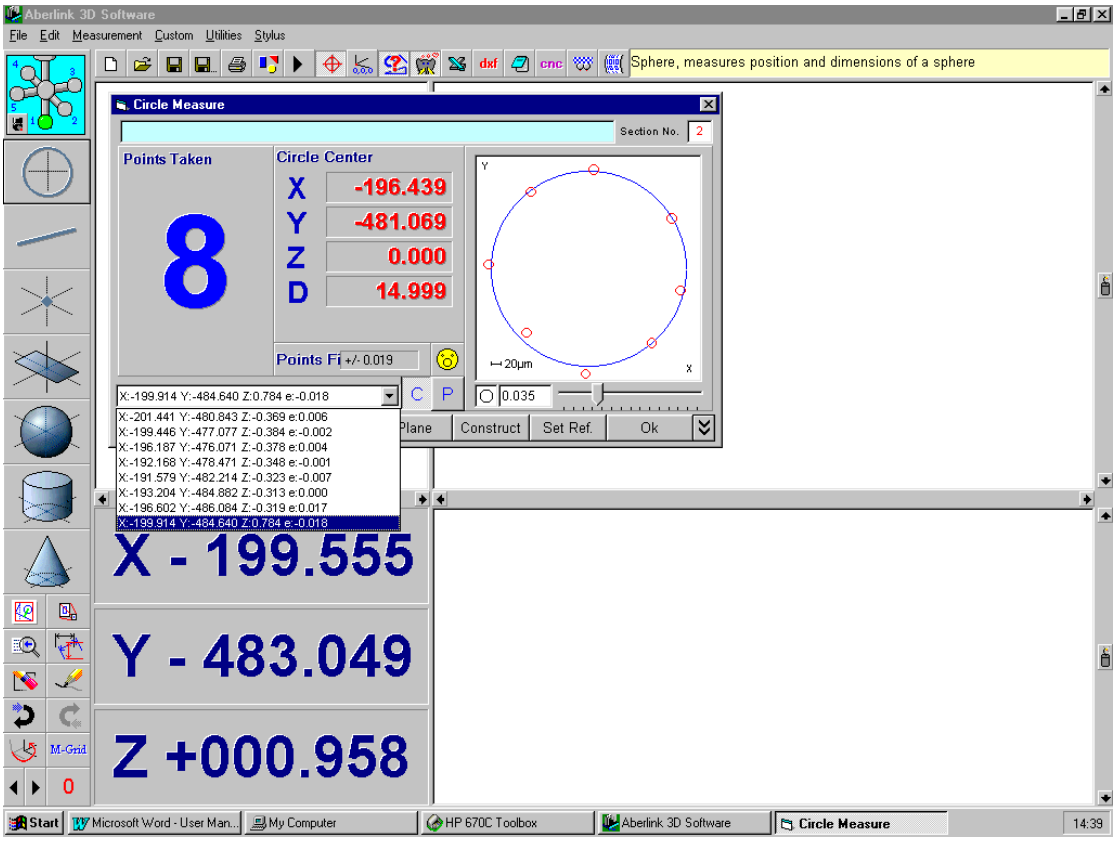
Dimensions can be called up between points within a Curve unit, and also geometric shapes can be constructed.

11.0 ERASING DATA POINTS

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. The methods for erasing data points are common to all measurement types.

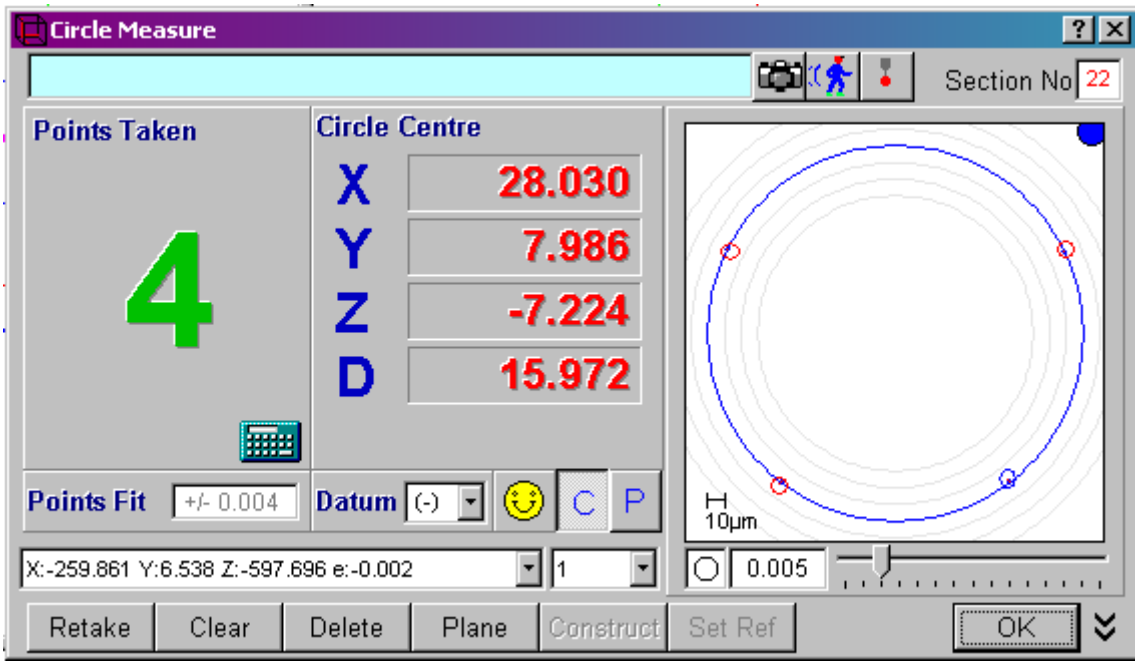
11.1 Retake

The ‘Retake’ button is used to erase a single measurement point. By default this will be the last taken point, however to erase a different individual point click on the down arrow situated just above the cancel button.



A list of the X,Y,Z values for all previously taken points together with an ‘e’ value representing the error of that point from the calculated best fit shape will now be displayed on a pull down menu. Highlight the desired point by clicking on it, and then click on ‘Retake’.

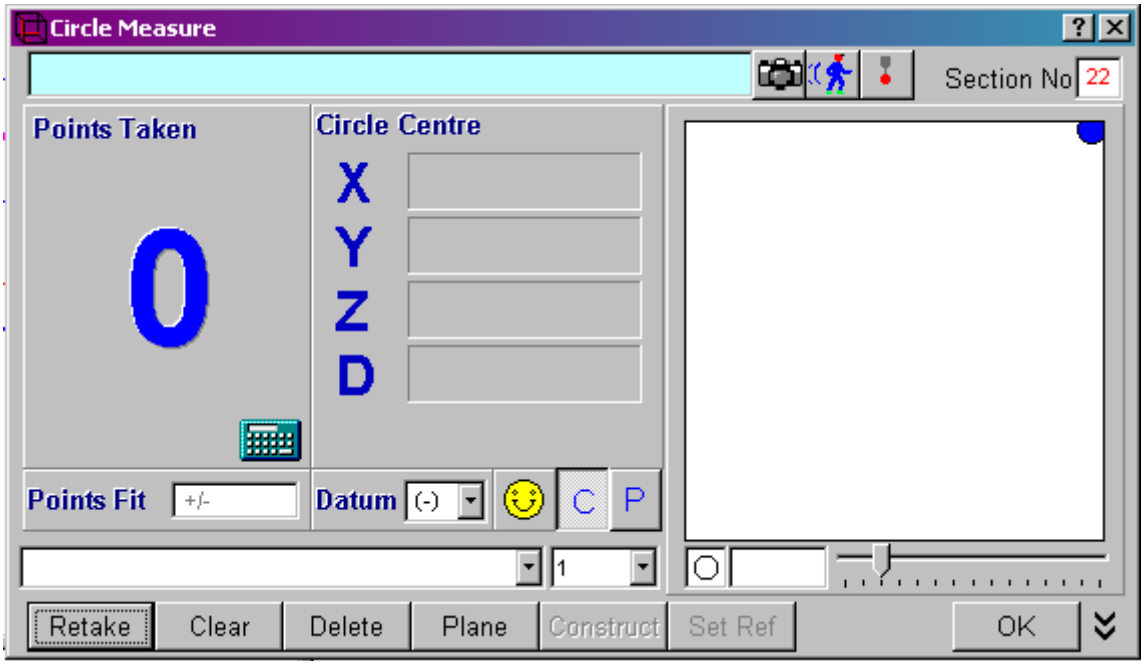
The screen will now recalculate the feature as if that measurement point had not been taken, ie. the number of points taken will reduce by one, and the erased point will no longer be shown on the graphical representation or taken into account in the displayed data.



It is also possible to right click on a specific point within the graphic window and select the ‘erase’ option.

11.2 Clear

To erase all the taken points on a particular measurement simply click on ‘Clear’. The number of points taken will now return to zero, and the measurement window will appear as when first opened.



11.3 Delete

Clicking on the delete button will erase all the taken points on a particular measurement and also close the measurement window and return to the main screen. This action is permanent and cannot be undone.

12.0 MANIPULATING THE GRAPHICS

During the inspection of a component, a pictorial representation of the part is built up on the screen in three 2-dimensional and one isometric view.

This chapter explains how to manipulate the graphics windows in the main screen. These techniques can be used to adjust the size and position of the graphical representation on the screen at any time during an inspection.

12.1 Zoom Out

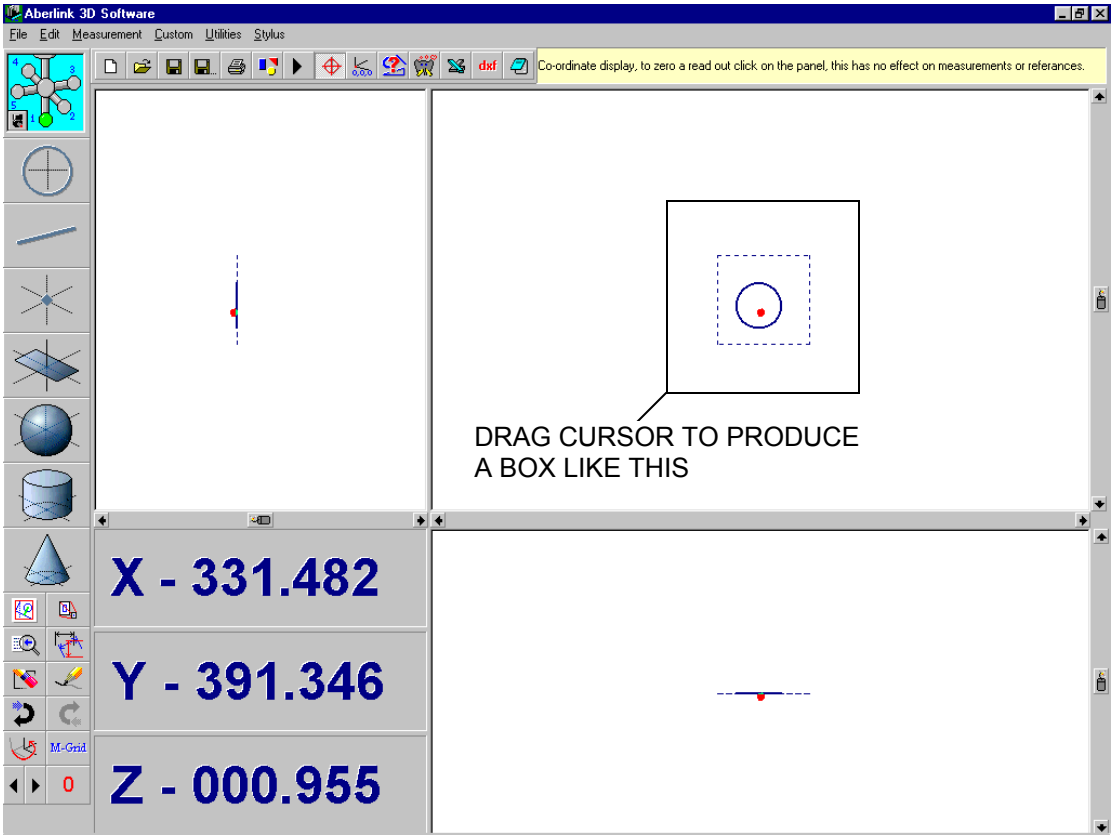
This button will make all 3 views zoom out by a fixed percentage.

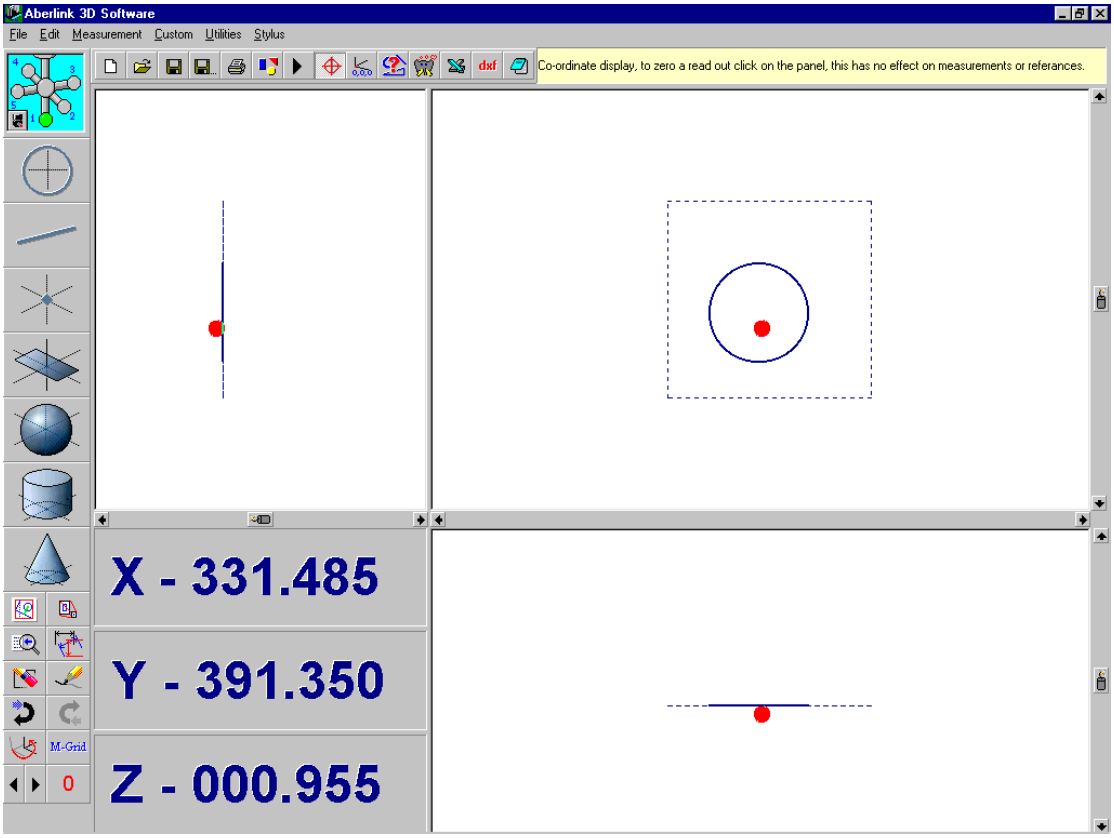
12.2 Zoom Full

This button will make all four views zoom out to make the entire measurement volume fill the screen. The edge of the measurement volume will be represented by a red rectangle.

12.3 Zoom In

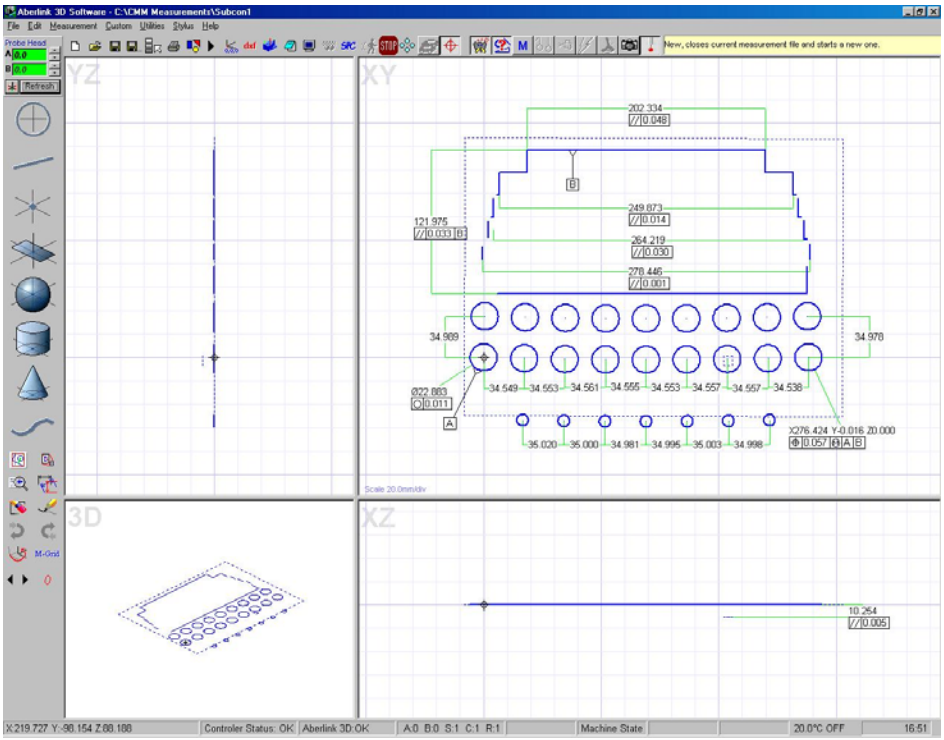
There is no button for zooming in. This is done by dragging the cursor, with the left mouse button depressed, to create a box around the desired area. When you take your finger off the mouse button the screen will zoom to the size of the box drawn.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
DRAG CURSOR TO PRODUCE
A BOX LIKE THIS
X - 331.482
Y - 391.346
Z - 000.955
M-Grid
0

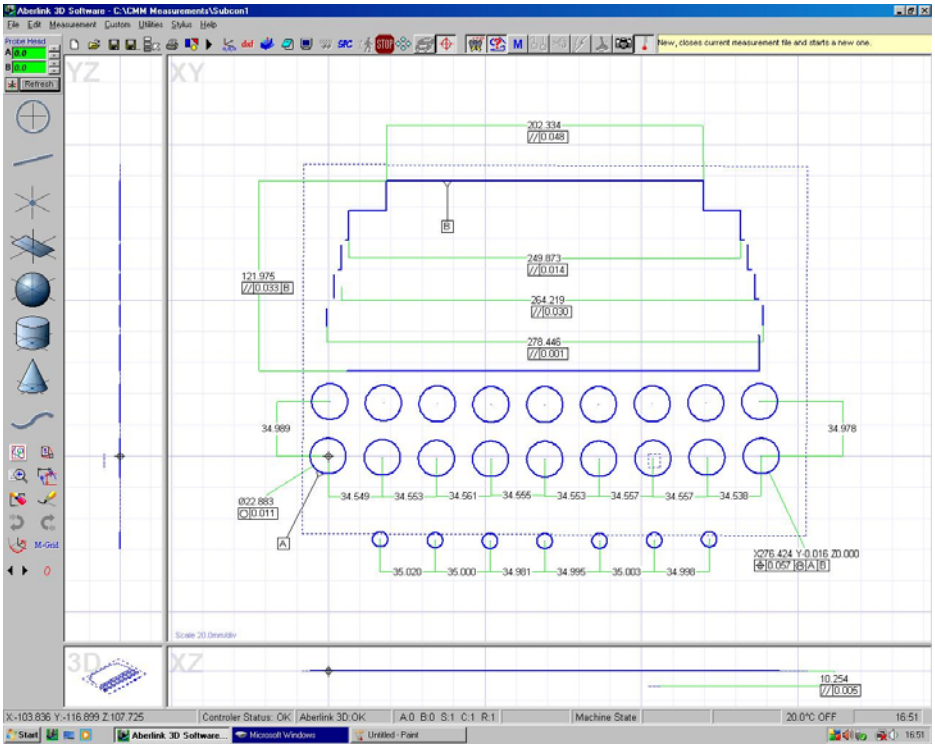

Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 331.485
Y - 391.350
Z - 000.955
M-Grid

12.4 Re-sizing & re-positioning the Graphics windows

It is possible to change the size and aspect ratio of the four views by clicking on the intersection between them and dragging it to the desired position on the screen.



Grab the intersection between the views and change as required:



12.5 Erase

To erase a feature or a dimension from a graphics window, simply click on the ‘erase’ button to highlight it, and then click on the feature that you want to erase.

Beware though, because the ‘erase’ button is latching, and you need to switch it off when finished, by clicking on it again.

12.6 Undo and Redo

If at any stage you make a mistake, for instance you have erased more features than you intended, you can click on the ‘Undo’ button, which will undo the last function performed. In this case the last erased feature will reappear. Similarly if you have undone a function by mistake, you can click on the ‘Redo’ button to bring it back.

12.7 Redraw

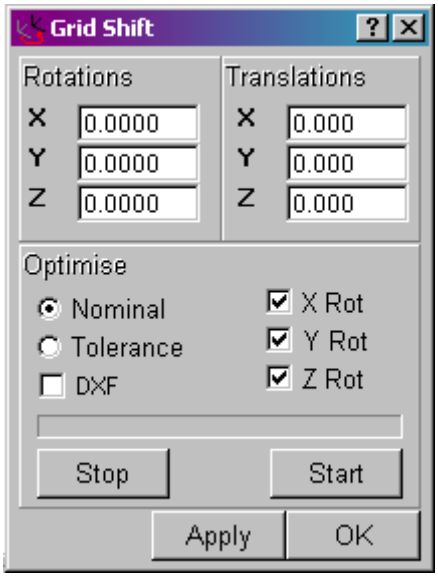
This is probably the most useful zoom button of all. When you click on this button the screen will automatically zoom to display all the measured features in all four views, and also refresh the screen view.

13.0 SHIFTING, ROTATING AND CANCELLING REFERENCES

13.1 Applying Rotations or Shifts

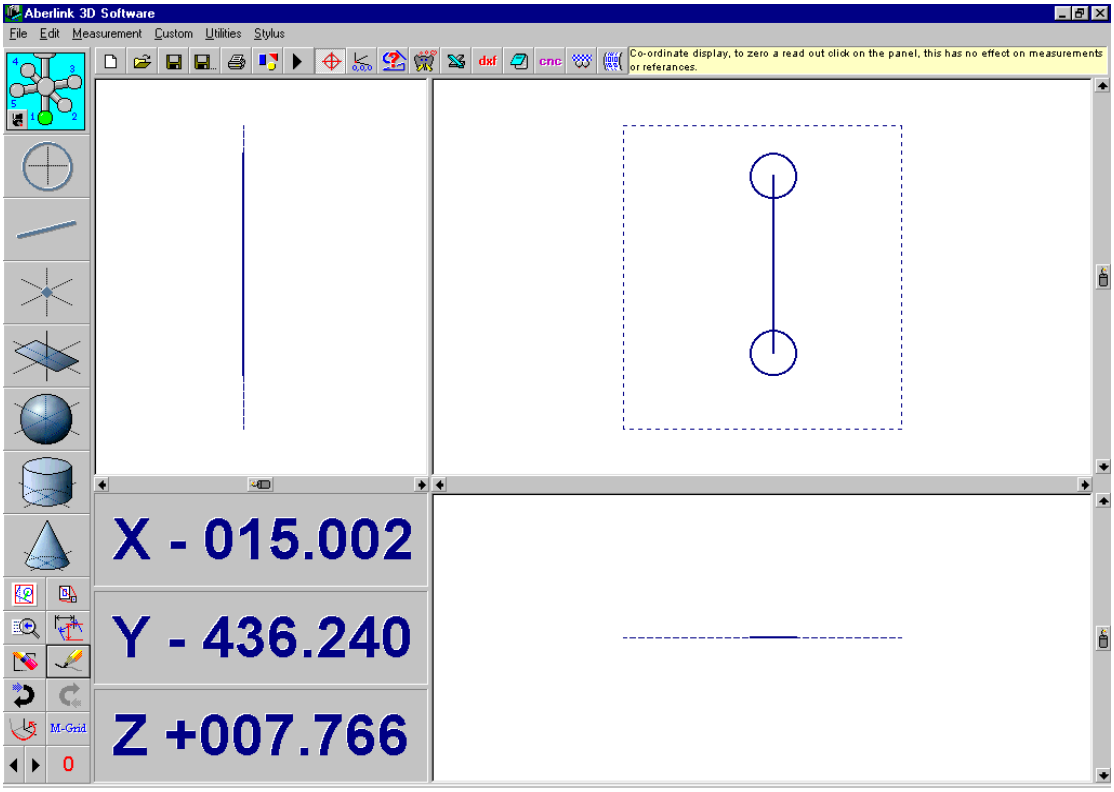
It may be necessary during an inspection to apply a rotation or shift to the inspected features. This can be achieved by using the Grid Shift button:

Clicking on this button will bring up the following window:

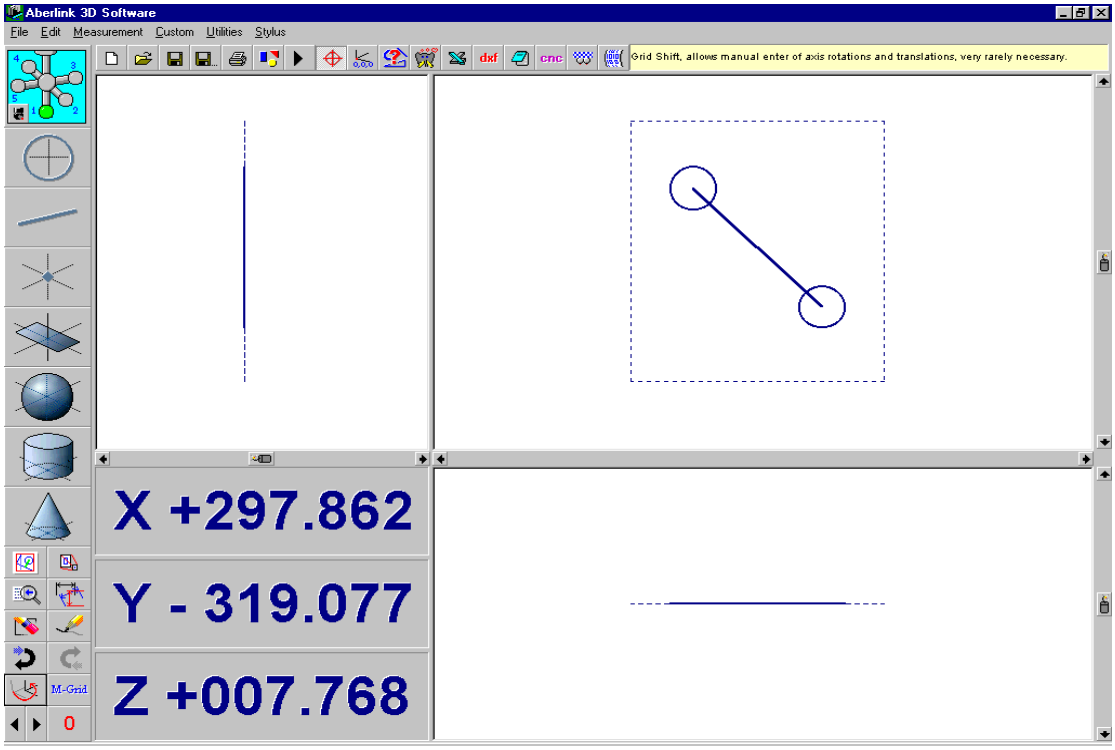


Now it is possible to apply a rotation or a translation to the measured features by typing the required angle(s) or distance(s) into the relevant box(es).

For instance, consider a component that consists of a plate with holes in its diagonal corners, which define a datum at 45 degrees. The best method for inspecting the plate is to measure the position of the 2 holes, construct a line between them and then set this line as a reference.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 015.002
Y - 436.240
Z +007.766
M-Grid
0

Now click on the grid shift button, and type in 45 in the rotation about Z box. Click on ‘Apply’ followed by ‘OK’.



The component will now be aligned at 45 degrees to the line between the two holes.

It is worth noting that when applying a rotation to inspection results, the rotation will be performed about the origin set at the time. Also if rotations are required about more than one axis, then the order in which the rotations will be applied is Z first, Y next and then finally about X.

Applying translations to measured features works in exactly the same way. Note however that the rotations and translations are non-associative, ie. If you wish to apply a combination of translations and/or rotations to measured features, then they must all be applied at the same time.

13.2 Machine Grid (Cancelling References)

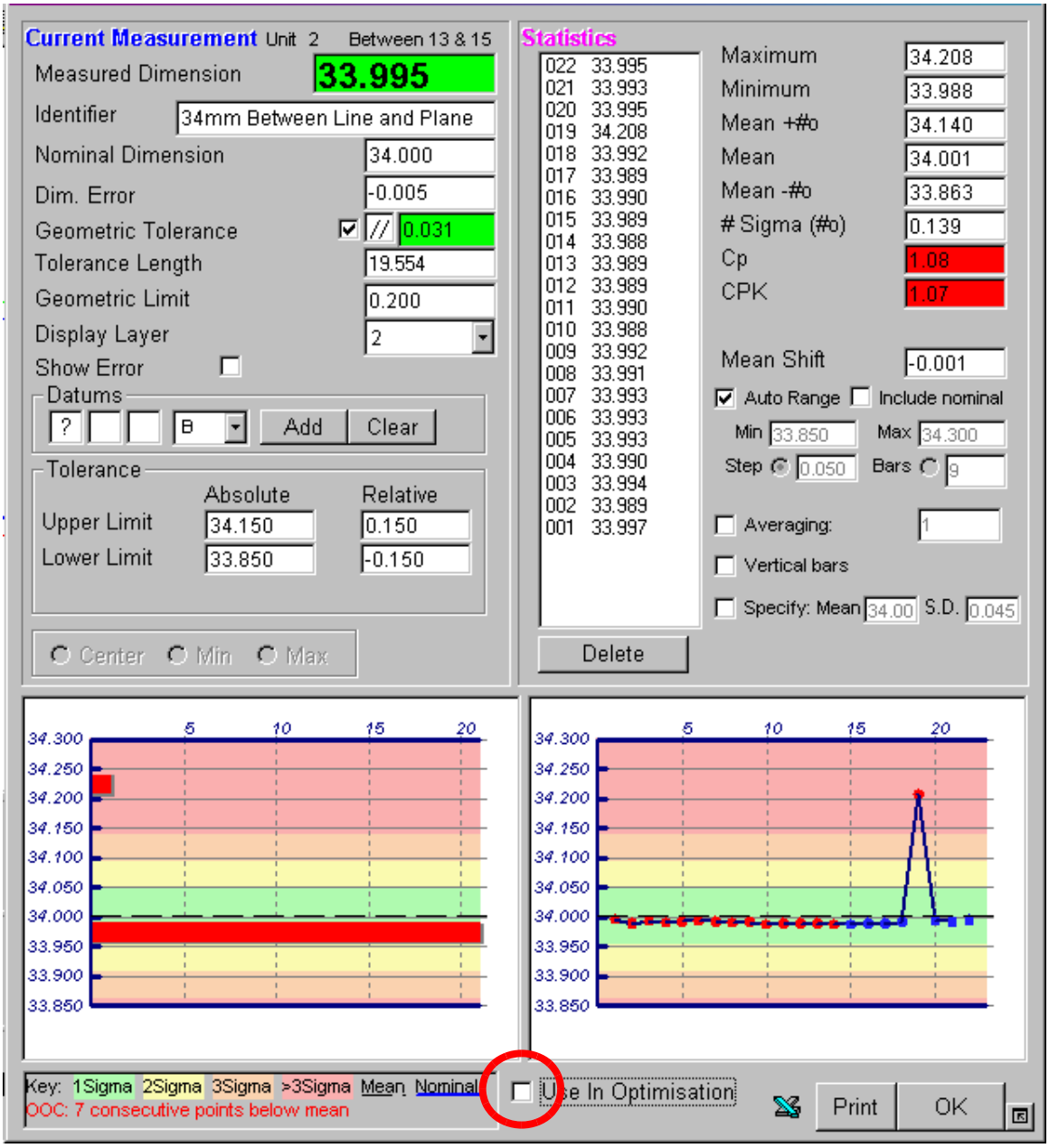
If at any point during an inspection you wish to cancel all the references that have previously been set, then click on the Machine Grid button

Having clicked on the M-Grid button you can now reset the alignment of the inspection by setting the required features as references.

13.3 Optimising Alignment

It is possible with some complex shapes that the optimum alignment for the component is not achieved by referencing any one, or series of features. In this instance it is possible to select certain dimensions on the measured component, and then optimise the alignment of the component in order to minimize the errors of these dimensions.

Firstly select the dimensions that you wish to use to optimise the alignment of the component, by right mouse clicking on the dimension to bring up the dimension detail window.



Change the nominal dimension to its correct value, if necessary, and then tick the box labelled ‘Use in Optimisation’. Click on OK to return to the main screen.

When you have selected all the dimensions that you wish to use in the optimisation, click on the Grid Shift button this will open the grid shift window, in the optimise section of the grid shift window you have two modes of dimension optimisation which you select by clicking on the small buttons next too Nominal / Tolerance.

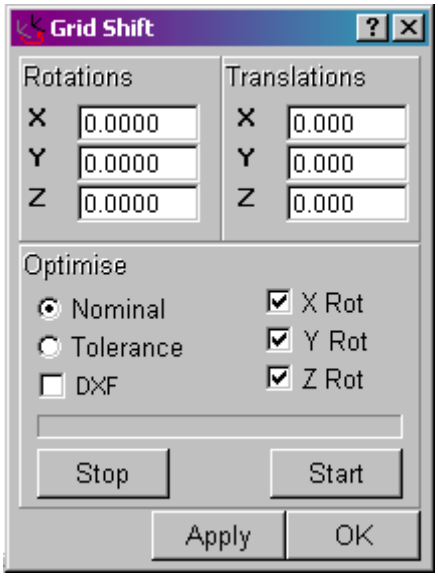
Nominal optimisation: If you select this option the software will translate & rotate the alignment system to get the optimum dimensions as close as possible to the nominal.

Tolerance optimisation: If you select this option the software will translate & rotate the alignment system to get the optimum dimensions within the upper & lower tolerance for that dimension.

Note If some of the dimensions are more important than others you can bias the optimisation by make their tolerance smaller, forcing any errors on to the other dimension.

It is also possible to optimise the alignment with out rotating the part in X,Y or Z. To do this you need to switch off the X,Y,Z rotation options in the optimum dimensions alignment window by click the ticks on or off in the 'tick boxes' next to X Rot, Y Rot, Z Rot

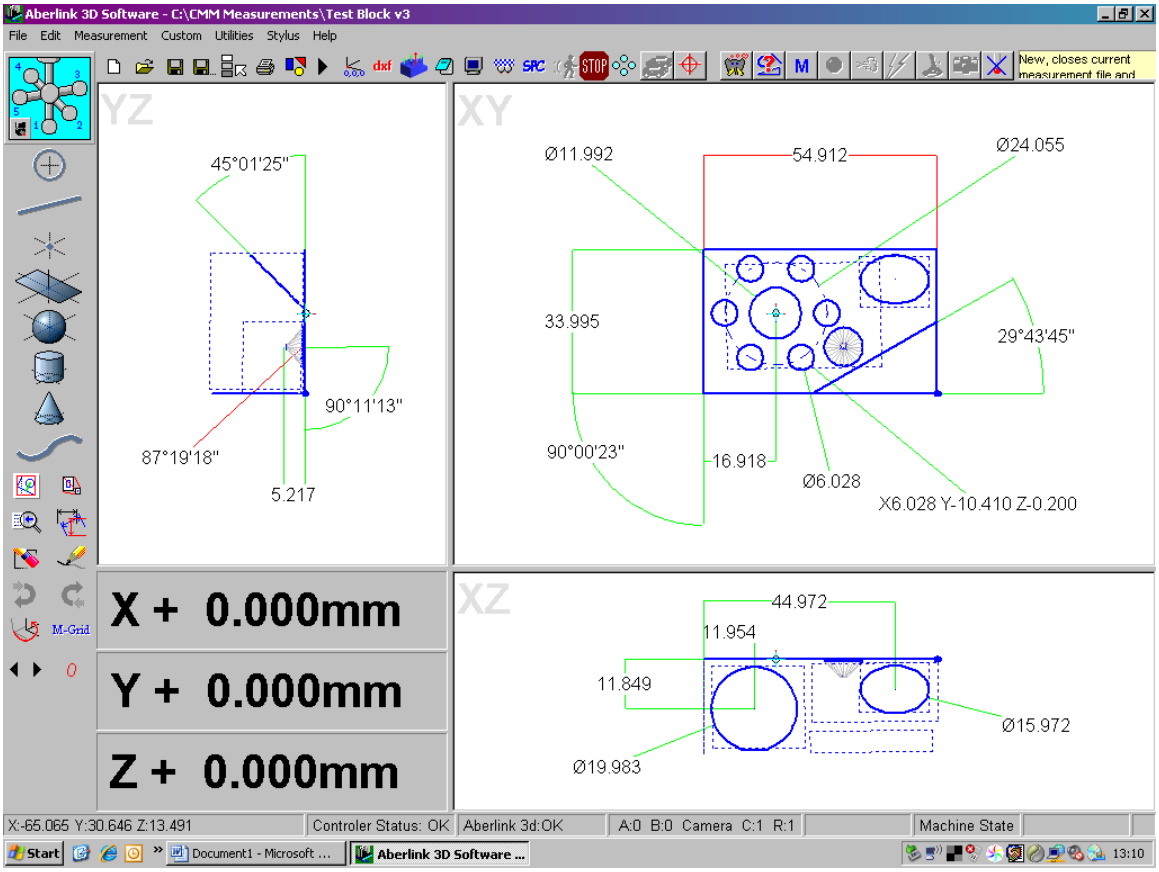
After you have select the option you want click on the 'start button'
The software will take a few seconds to calculate the best alignment, and when it has finished click on OK.



Note that the minimum number of dimensions should be selected that best define the rotation required for the component, ie. Typically this will be one dimension for X, one for Y and one for Z. This will help to minimize the calculation time required.

14.0 BRINGING UP DIMENSIONS ON THE SCREEN

One of the features of the Aberlink 3D software is that the dimensions for the measured component can be brought up on it's graphical representation, in the same way as they are shown on the component's engineering drawing.



To bring up the dimensions onto the screen all you have to do is select the features that you wish to measure between, by clicking on their graphical representation. If the software doesn't default to the type of measurement that you want (horizontal, aligned etc.), then you will also have to select the correct dimension type.

In most cases the order in which you select the features will make no difference. The instances in which the order of selection does make a difference are highlighted later in this chapter.

It is important to grasp the following principles when bringing up dimensions:

1. Circles and spheres are treated as single points in space, with all dimensions being to their centre.
2. Cylinders are treated as 3 dimensional lines along their axis. When dimensioning between 2 lines (or cylinders etc.) the order of their selection is important.

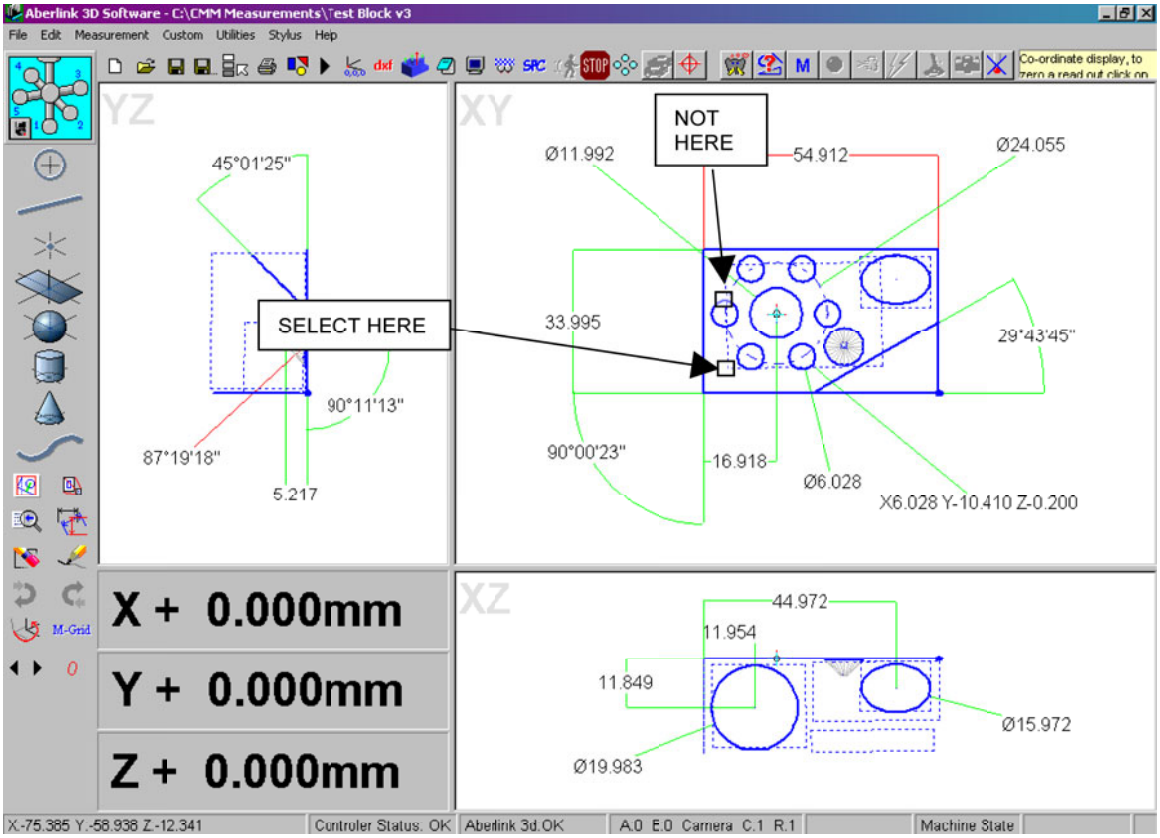
3. Cones may be treated either as a line along it's axis, or as a point at it's apex, depending on whether the cone is selected as the first or second feature.

4. When linearly dimensioning a feature to a plane, the dimension will be perpendicular to the plane. When dimensioning between 2 planes the order of their selection is important. (see section 16.5 – Dimensioning between two Lines or Planes)

14.1 Selecting Features

When bringing up dimensions on the screen, it is necessary to select the features that you wish to dimension in the graphical representation.

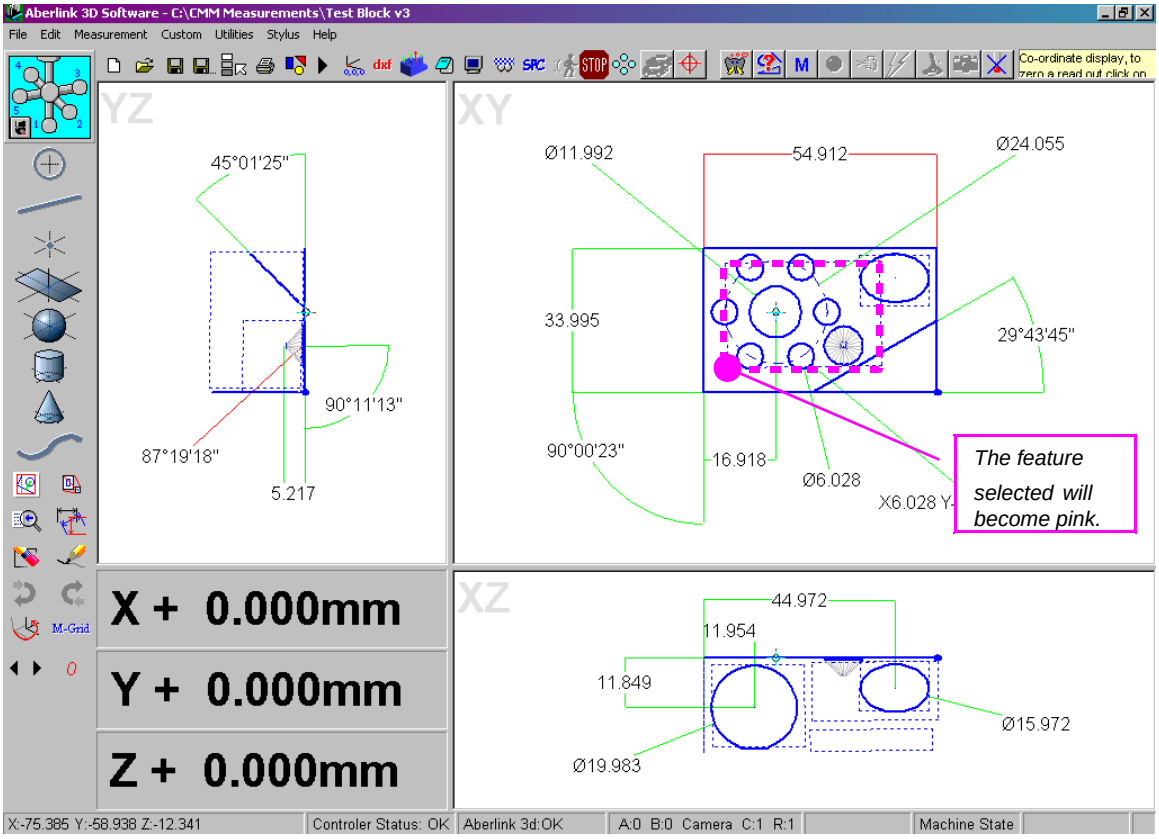
To select a feature, click on the dark blue outline of the feature. Make sure that at the point that you click on the feature, there are no other overlapping features within the selection box.



Note – You cannot select a 3 dimensional feature by clicking on the grey lines that represent its depth.

Also Note – If you are dimensioning in the side view of a reference plane, which has had 2 dimensional features projected into the plane, then a single line on the graphics will represent both the plane and the features. In this instance clicking on the line may select any one of the features. To define the specific feature required you must use the feature select buttons.

The feature selected will become pink.



If you have trouble selecting a feature there are various courses of action.

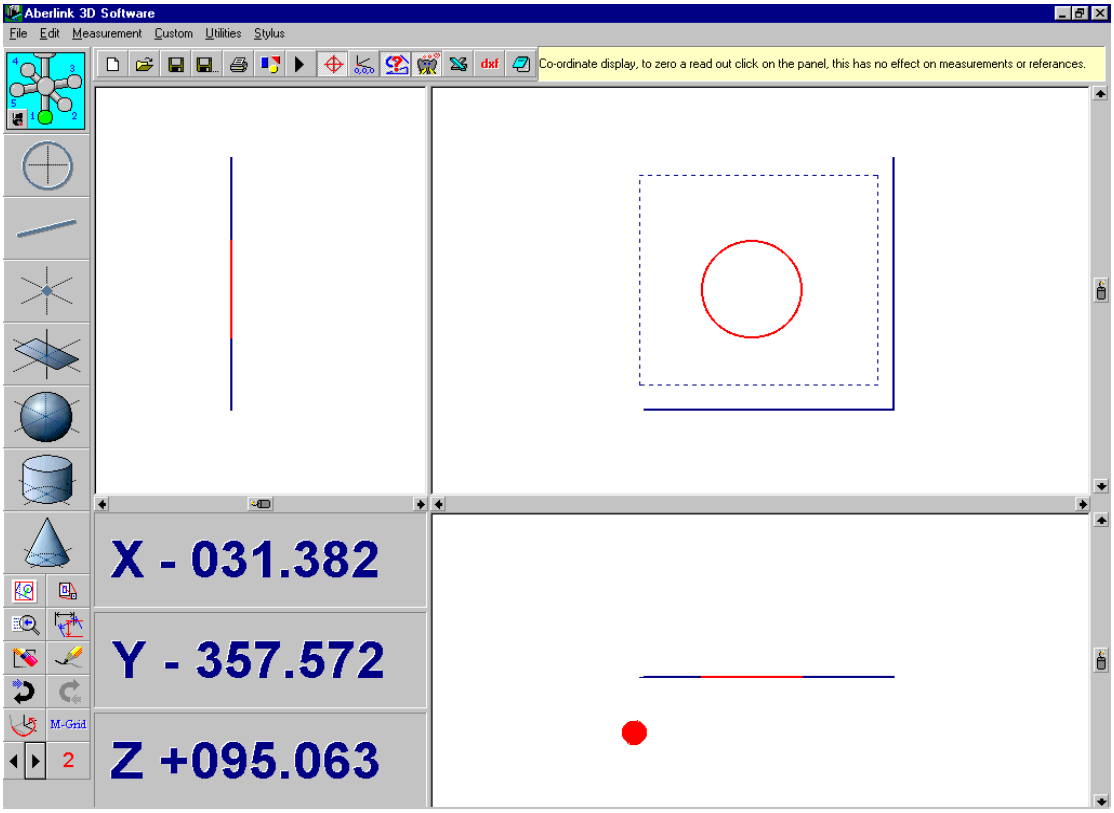
1. You can use the feature select button to assist selection of the feature.
2. You can zoom in more closely onto the feature, to make it easier to select.

14.2 Using the Feature Select Buttons

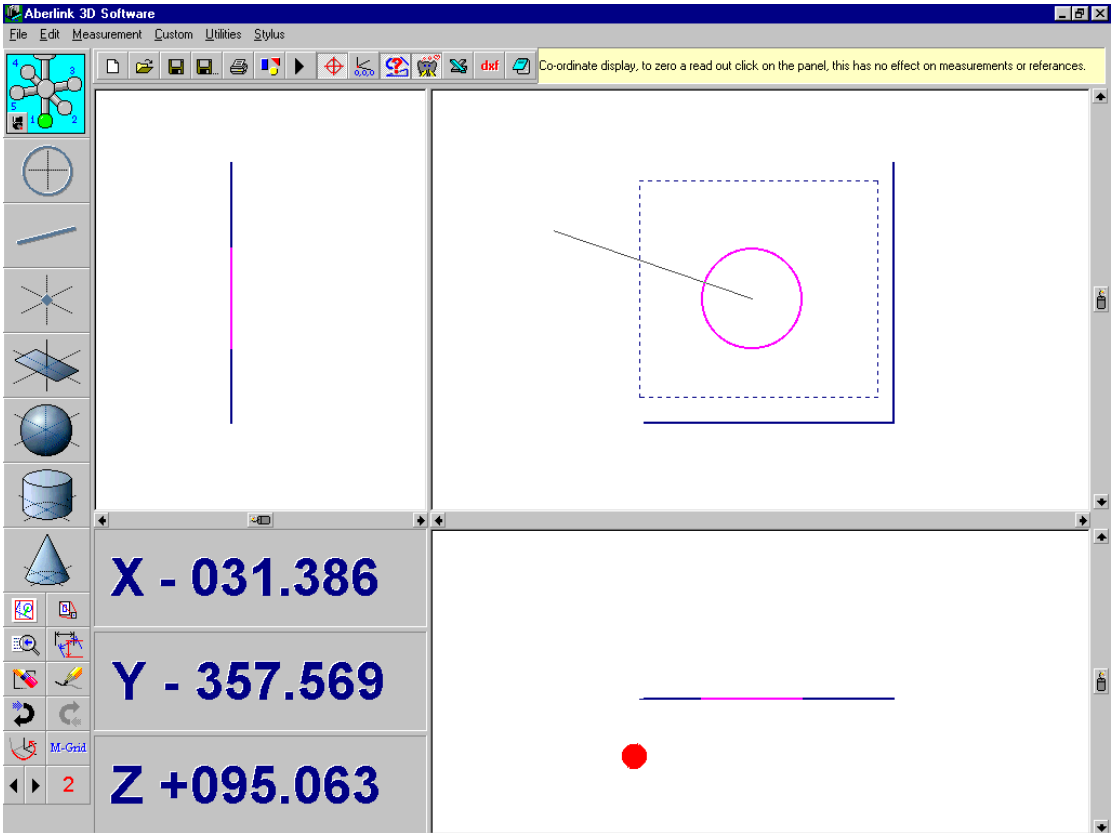
In some instances it will be difficult to select a feature by clicking on it directly. For instance, the same line will represent both the side view of a circle, and also the plane into which it is projected. If you have difficulty in selecting a feature, the Feature Select buttons will provide the solution.

The red number (initially zero) refers to the feature number in the order that it was measured. For instance if you measured a plane first, followed by a circle and then a line, the plane would be feature number 1, the circle number 2 and the line number 3. Now you can use the right and left arrows to increment and decrement the feature number, to obtain the feature desired.

As the feature number is selected, the graphical representation of that feature will become red.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 031.382
Y - 357.572
Z +095.063
M-Grid
2

Now clicking anywhere within the Main Screen will select the desired feature (even when clicking directly on another feature).


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 031.386
Y - 357.569
Z +095.063
M-Grid
2

The Feature Select button will only work to select a feature once. If you need to select the same feature twice (for instance to display the diameter of a circle), simply click on the red number again to re-highlight that feature. (This removes the necessity to use the right and left arrows to move off, and return to the desired feature). Now click on the Main Screen again to re-select the feature.

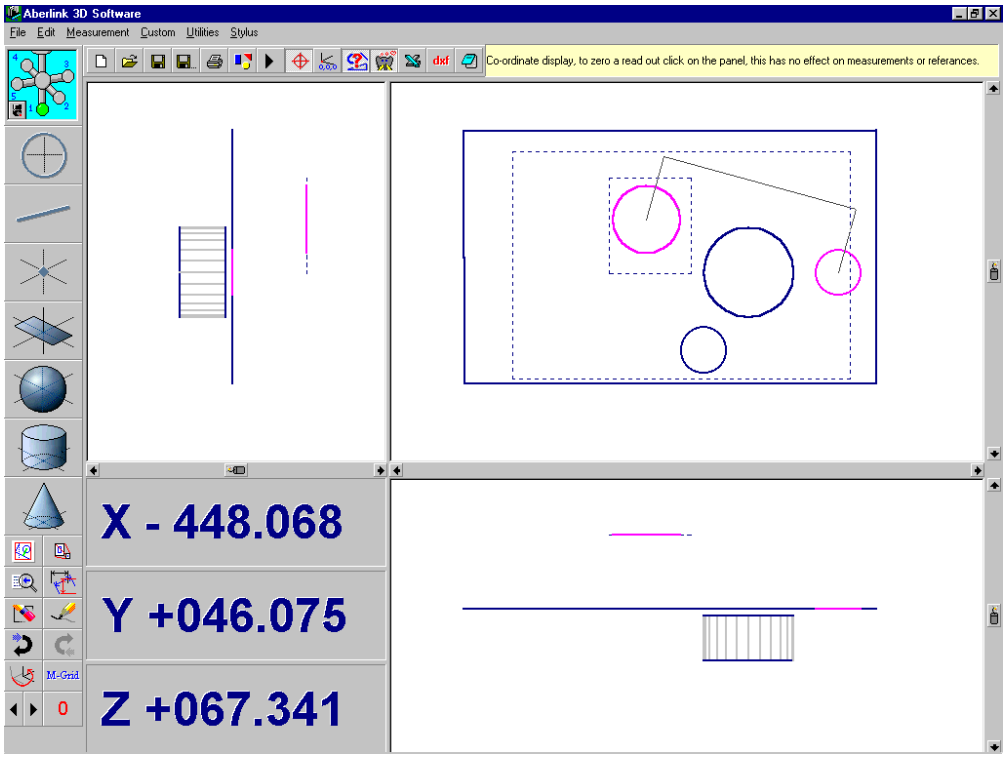
14.3 Aligned Dimensions

14.3.1 Three Dimensional

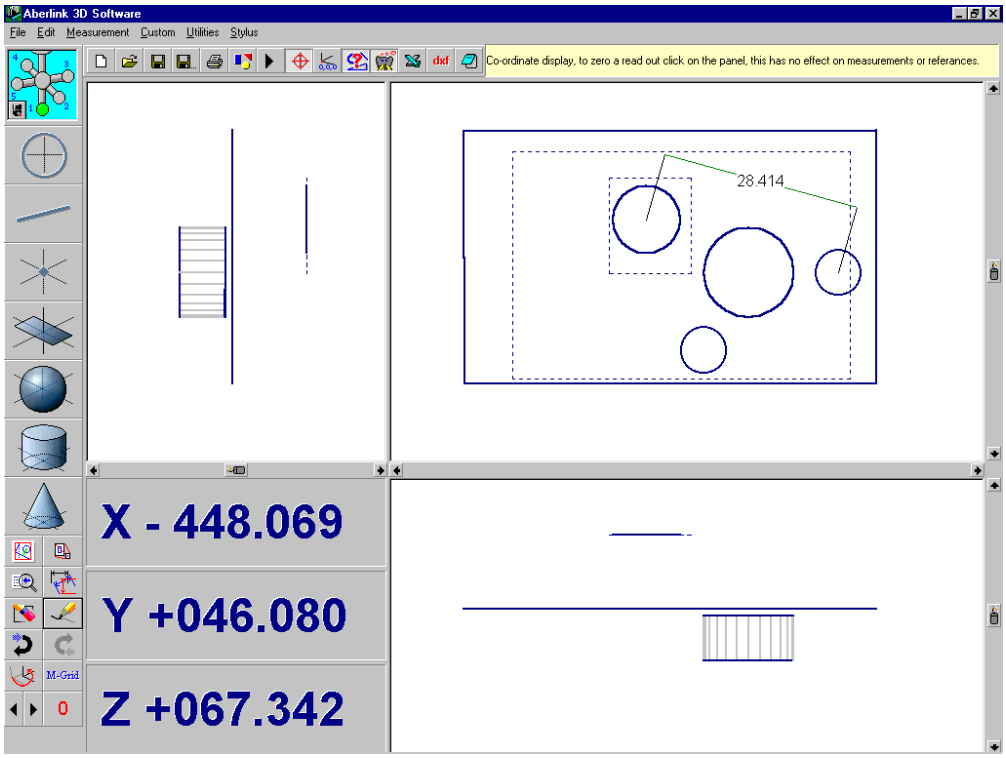
An aligned dimension can be described as either the shortest distance between two points in space, or as the perpendicular distance between a line and a point. If the 2 features being dimensioned are not in the same plane then this will produce a 3 dimensional aligned dimension.

When selecting two points (or circles or spheres) to dimension between, or a line and a point (or equivalent) the software will automatically default to producing a 3 dimensional aligned dimension.

Now select the second feature to which you wish to dimension. Again, if correctly selected the second feature will also turn pink, and the outline of some aligned dimension leader lines will appear.

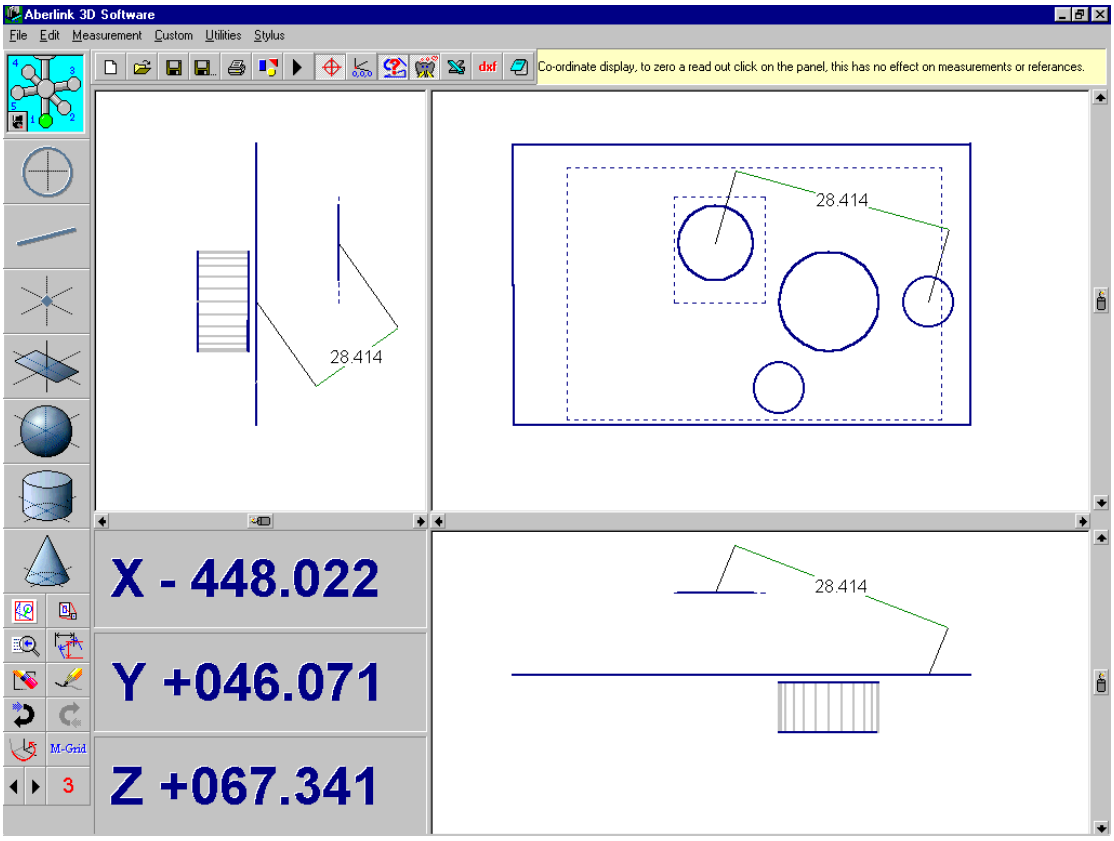

Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 448.068
Y +046.075
Z +067.341
M-Grid
0

These leader lines can be positioned at any length on either side of the features by moving the cursor on the screen. When they are at a convenient position click for a third time. The aligned dimension will now appear.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
28.414
X - 448.069
Y +046.080
Z +067.342
M-Grid
0

Note – If when trying to select the second feature you click on blank screen by mistake, (ie. the second feature is not selected correctly) the measurement will be aborted, and the first feature that was selected will return to it's normal dark blue colour. This can be a useful technique for aborting a measurement where the first feature has been selected in error.

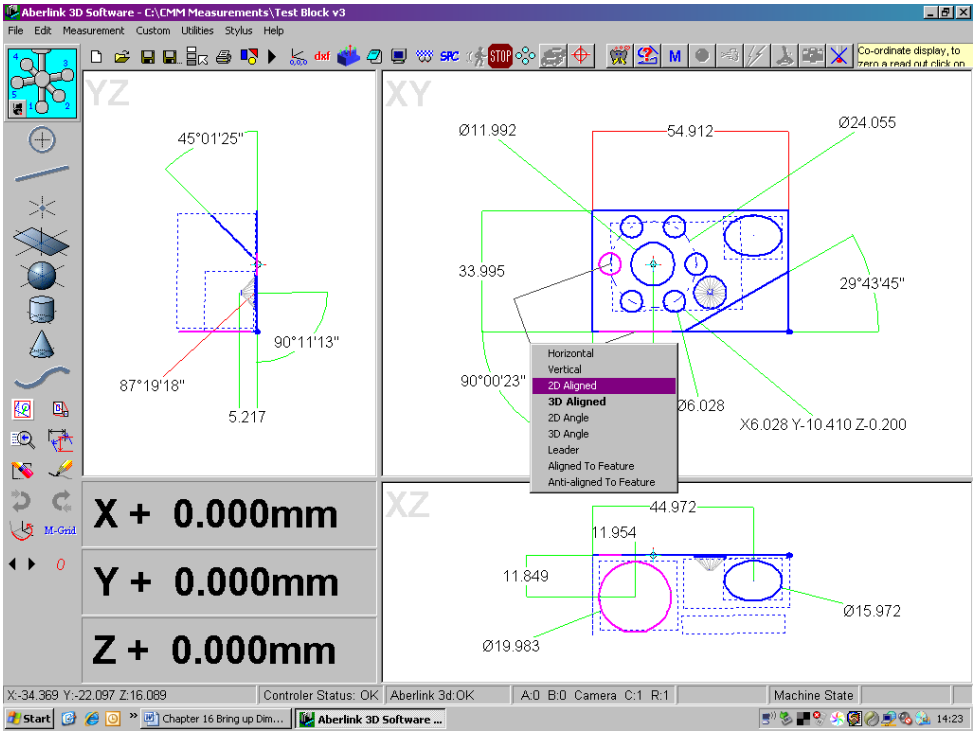
Note that as the two circles being dimensioned are in different planes, the aligned dimension is 3-dimensional. This is best shown by bring up the same dimension in the other two views.



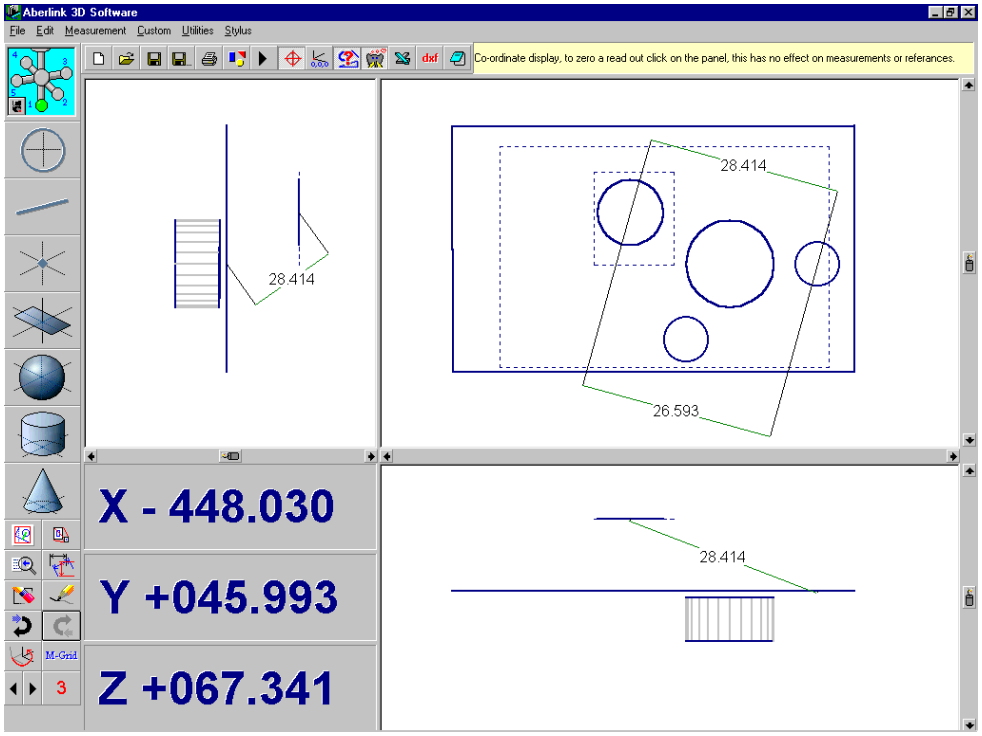
14.3.2 Two Dimensional

It is possible to force aligned dimensions to be aligned only in the 2 dimensions of the view in which it is called up. For instance perhaps you are interested only in the X,Y separation of the 2 holes dimensioned above.

In this case select the 2 circle features as before, but this time before placing the dimension, right click on the Main Screen. The following menu will appear:



Select the 2D aligned option and you can now position the dimension as before.



Also note -that the 2D and 3D aligned dimensions will appear identically in the 2 dimensional views, although having different values. Care should therefore be taken when dimensioning between features that are not in the same plane.

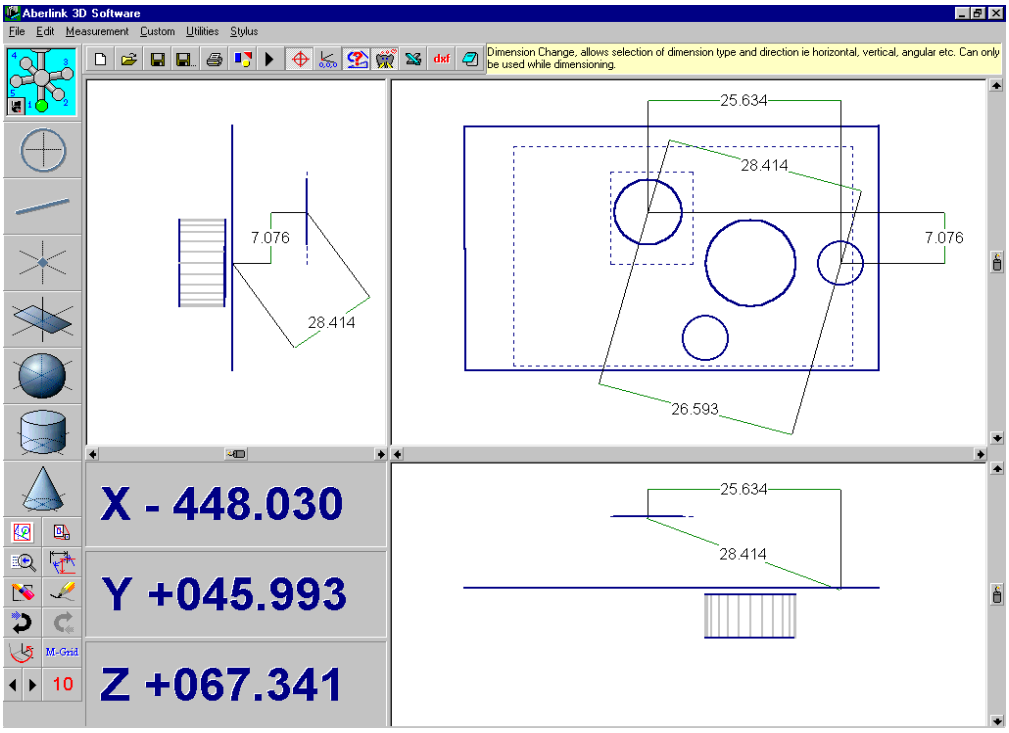
14.4 Horizontal and Vertical Dimensions

A horizontal or vertical dimension can be described either between two points in space, or between a line or plane and a point, in which case the dimension will be attached to the mid-point of the line or plane. Instead of being aligned between the features, these measurements will be perpendicular to the alignment of the component (which is defined by setting references. If no reference has been set, then the axes of the machine will be used.

> ***Note*** *– If the alignment of the component has not been defined, and the horizontal and vertical dimensions are perpendicular to the axes of the machine, these measurements will have little or no significance.*

When selecting two points (or circles or spheres) to dimension between, or a line and a point (or equivalent) the software will automatically default to producing an aligned dimension. A horizontal or vertical dimension must be defined as follows:

Select the features between which you wish to dimension, as for aligned dimensions above. The features will turn pink and the outline of some aligned dimension leader lines will appear. Now before positioning the dimension, right click on the Main Screen, which will bring up the Dimension Type menu. The 3D aligned dimension type will be highlighted by default, but click on the horizontal or vertical type button as required, and this button will now become highlighted and the software will return to the main screen with the outline of the dimension leader lines now showing the horizontal or vertical dimension, as selected.



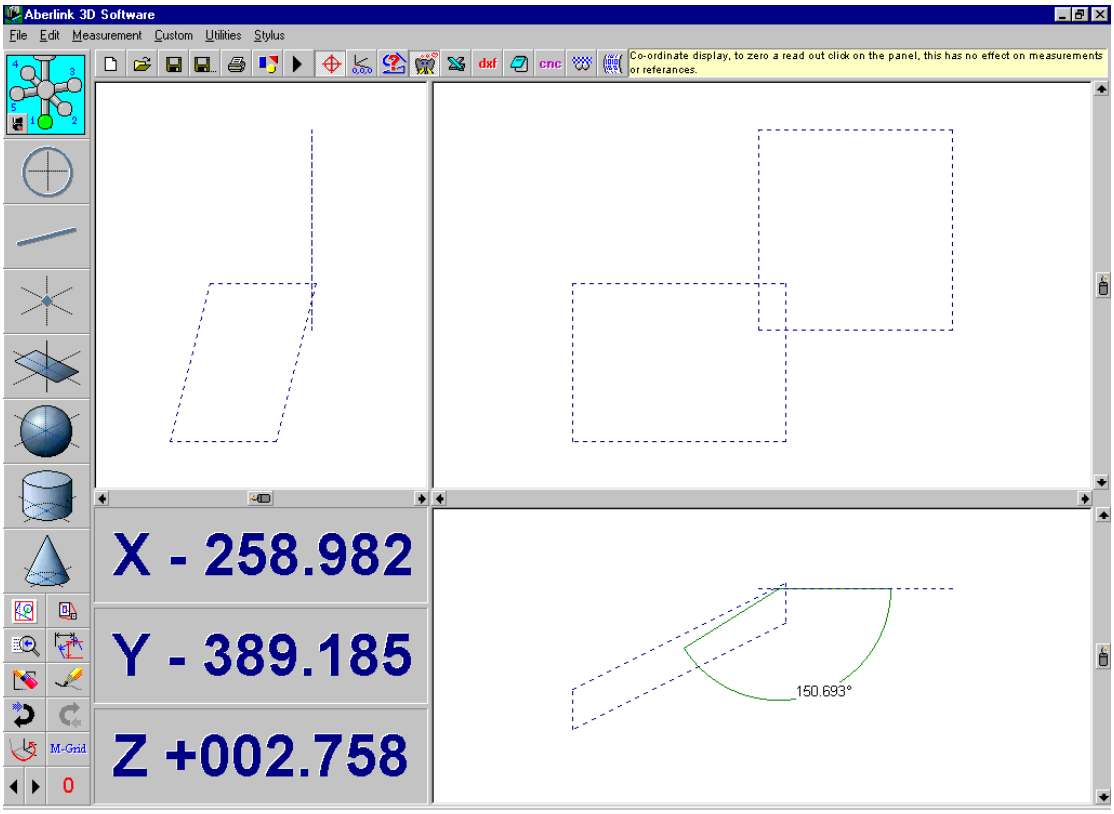
These leader lines can be positioned at any length on either side of the features, as for aligned dimensions, by moving the cursor on the screen. When they are at a convenient position click the mouse for a third time and the dimension will now appear on the screen.

14.5 Angular Dimensions

14.5.1 Three Dimensional

Angular dimensions may be created between two lines (or equivalent), or two planes, or between a line and a plane, or equivalent 3D shapes.

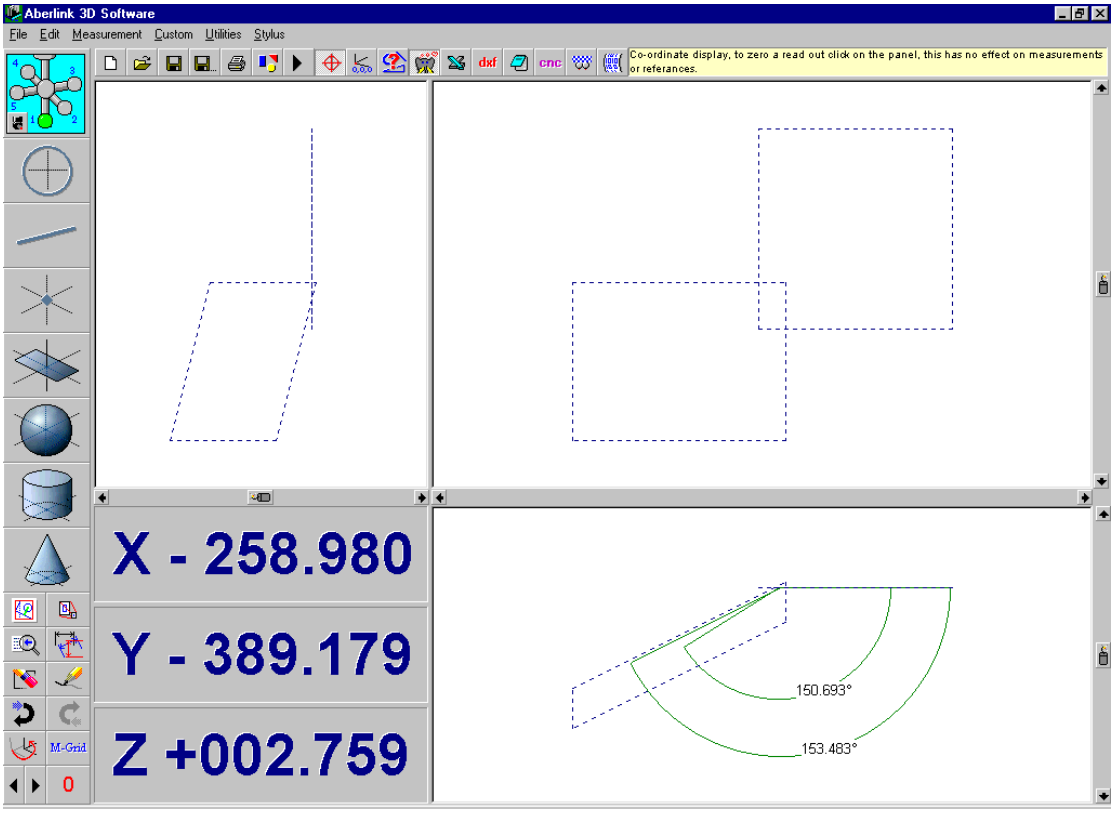
Select the features between which you wish to dimension, as for aligned dimensions above. The features will again turn pink if correctly selected. The software will default to an angular dimension and the outline of the appropriate dimension leader lines will appear on the screen. The leader lines can be positioned at any length in any quadrant of the angle, by moving the cursor on the screen. When they are at a convenient position click for a third time and the dimension will now appear on the screen.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 258.982
Y - 389.185
Z +002.758
150.693°

14.5.2 Two Dimensional

It is possible to split angles into the 2 dimensional components of the view in which it is called up. For instance perhaps you are interested only in the X,Y component of the angle between the 2 planes dimensioned above.

If this is required, having selected both features right click on the Main Screen and select 2D angle from the dimension type menu.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
X - 258.980
Y - 389.179
Z +002.759
150.693°
153.483°

Again note that the 2D and 3D angle dimensions will appear identically in the 2 dimensional views, although having different values.

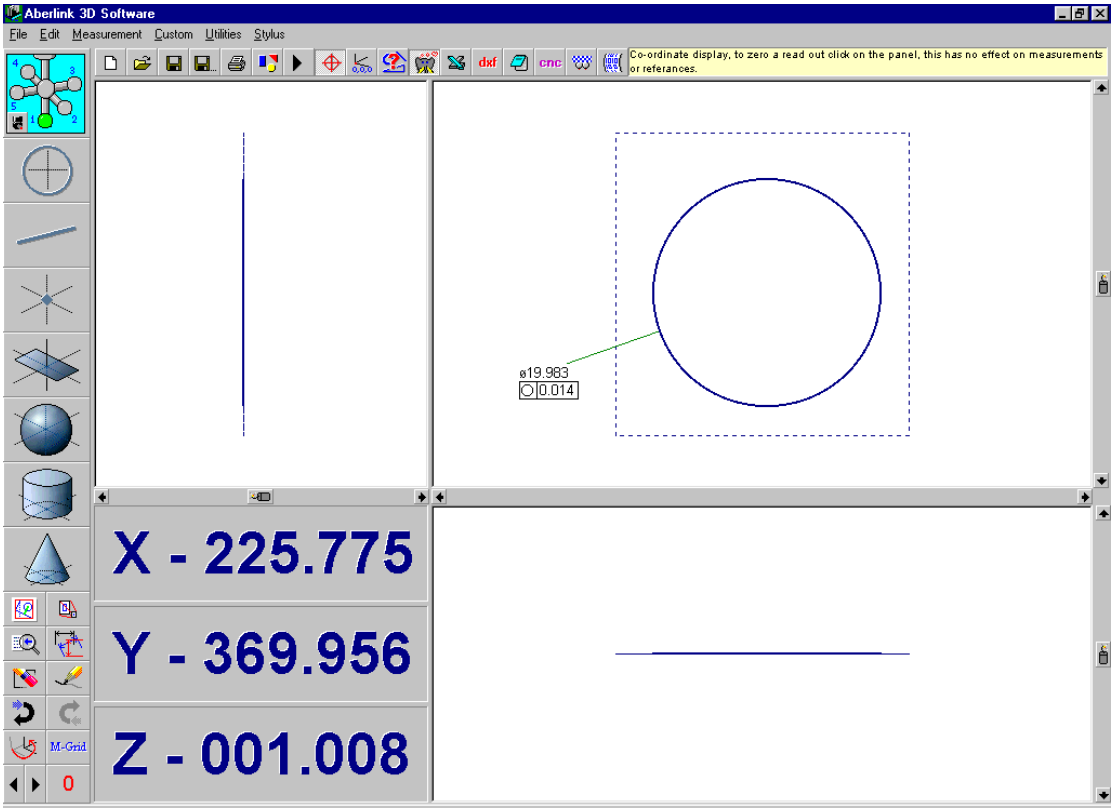
14.6 Diameter Dimensions

The diameter of a circle, cylinder or sphere may be brought up on the screen as follows:

Select the circle, sphere or cylinder as previously described.

> ***Note*** *– You cannot select a cylinder by clicking on the grey lines that indicate its depth.*

Now select the same feature again. The outline of a single dimension leader line will appear on the screen, and this can be positioned anywhere on the screen by moving the cursor. When it is at a convenient position, left click the mouse for a third time and the diameter will now appear on the screen.

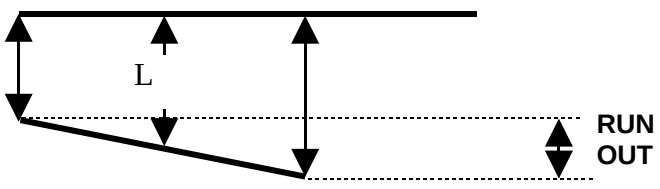

Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or referances.
ø19.983
0.014
X - 225.775
Y - 369.956
Z - 001.008
M-Grid
0

14.7 Dimensioning between two Lines or Planes

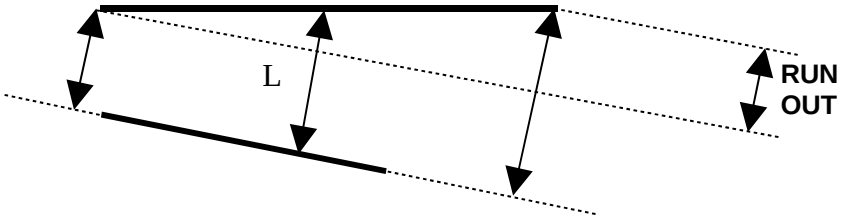
A linear dimension may be produced between two lines or planes by attaching the dimension to the first line or plane selected, and extending it either perpendicularly (for an aligned dimension), or horizontally or vertically (for horizontal and vertical dimensions), until it reaches the mid-point of the second one.

If the lines or planes are not parallel, different dimensions will be achieved depending on which line or plane is selected first. Also the value for run out is calculated by projecting perpendicular construction lines from the first selected line, to meet the ends of the second line. The run out (or perpendicularity) will be the difference between these two values.

A) Selecting the long line first:



B) Selecting the short line first:

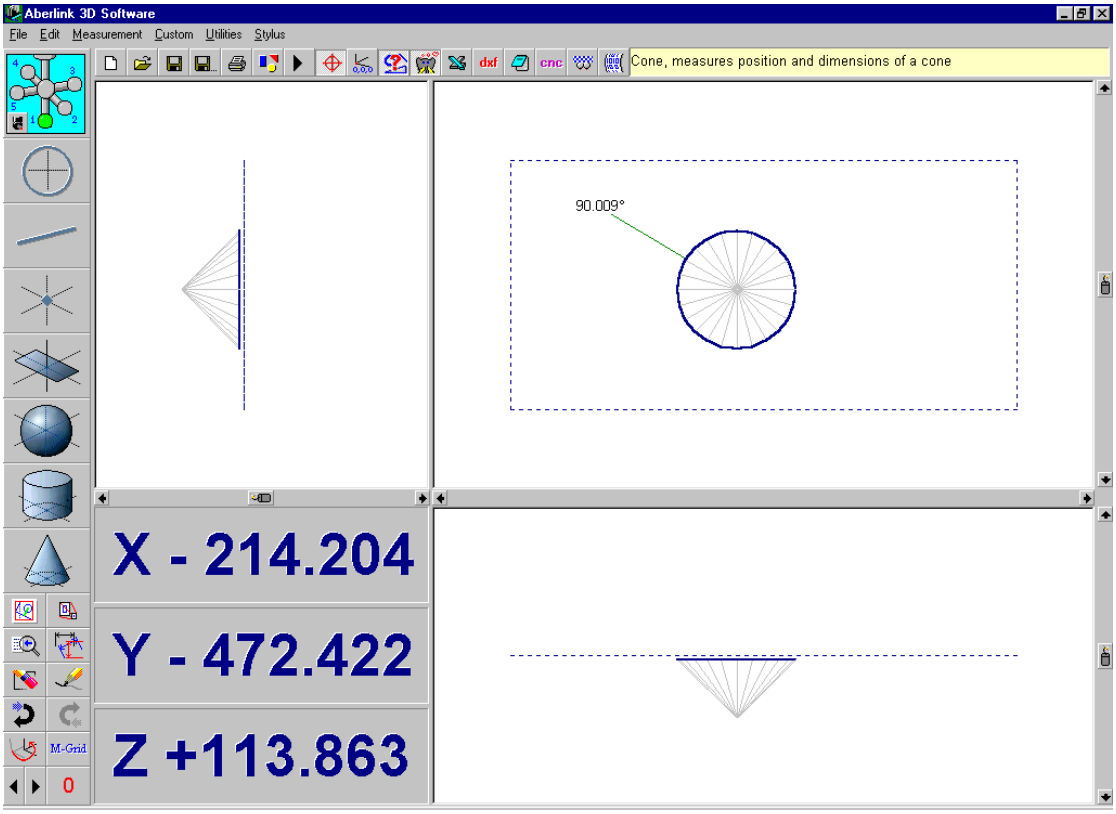


The first line selected, in effect, becomes the reference line for the measurement. If you are measuring the distance between a long line and a short line it will be better to select the long line first (unless the short line is specifically defined as a datum).

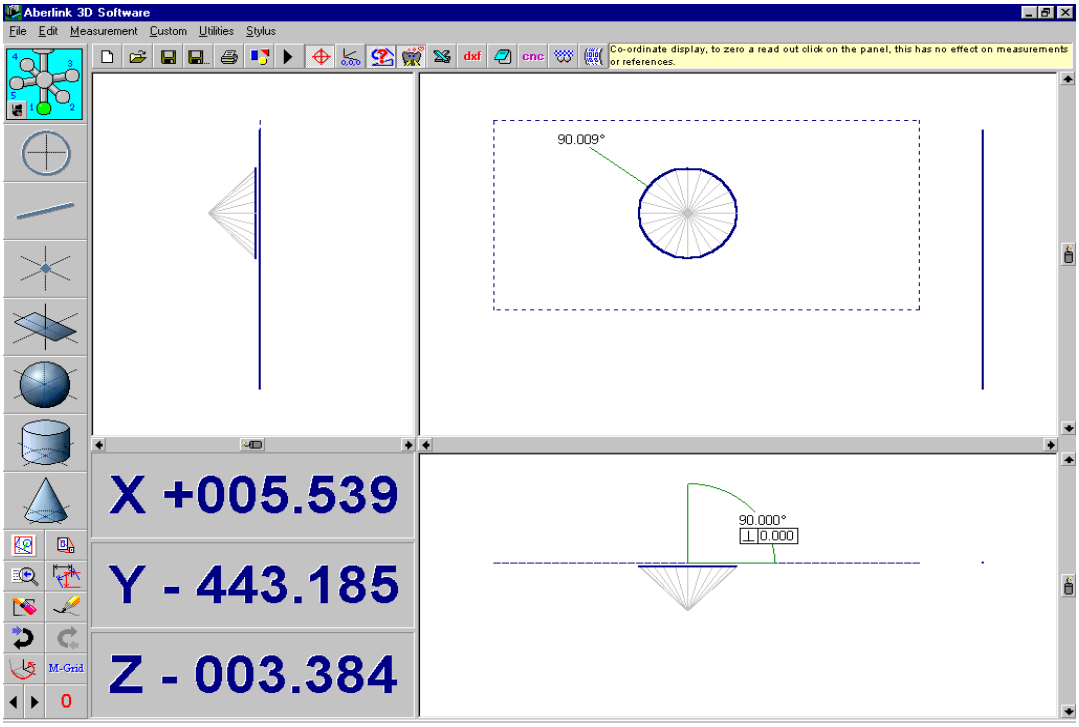
14.8 Dimensioning to a Cone

A cone is a special case, as when measuring to it, it may be treated either as a line along it's axis, or alternatively as a point at it's apex.

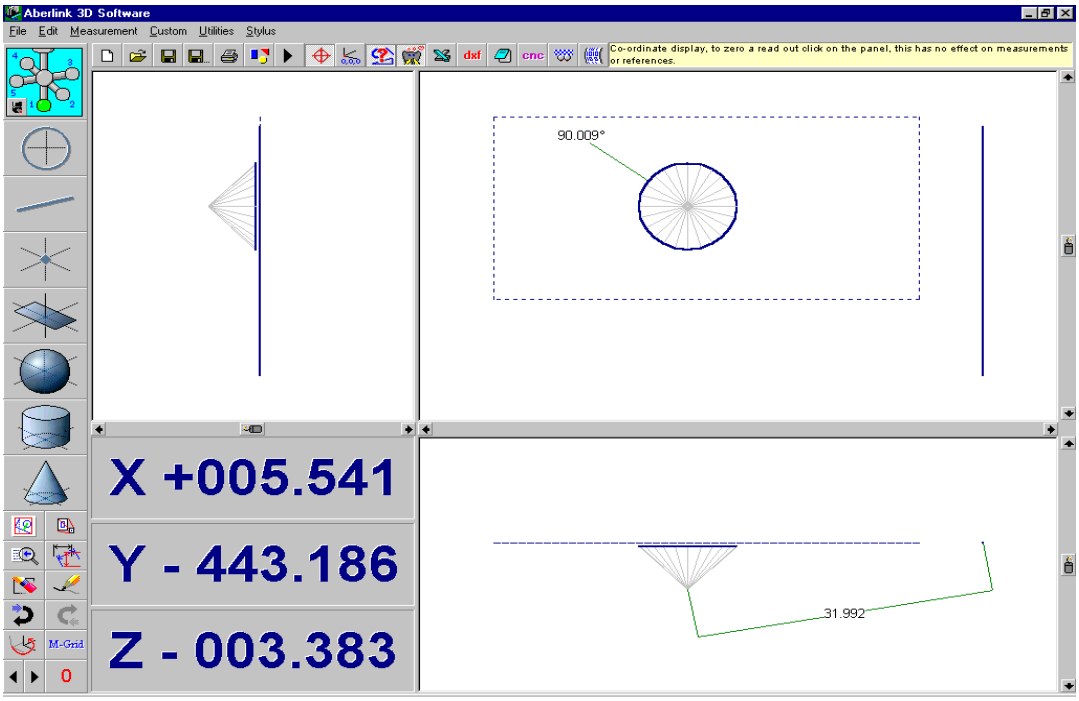
Selecting a cone twice will display it's included angle.



When bringing up dimensions to a cone, the cone is treated either as a line or a point, depending on whether it is selected as the first or second feature. If selected first, the software will treat the cone as a line along its axis, and produce a relevant dimension accordingly.

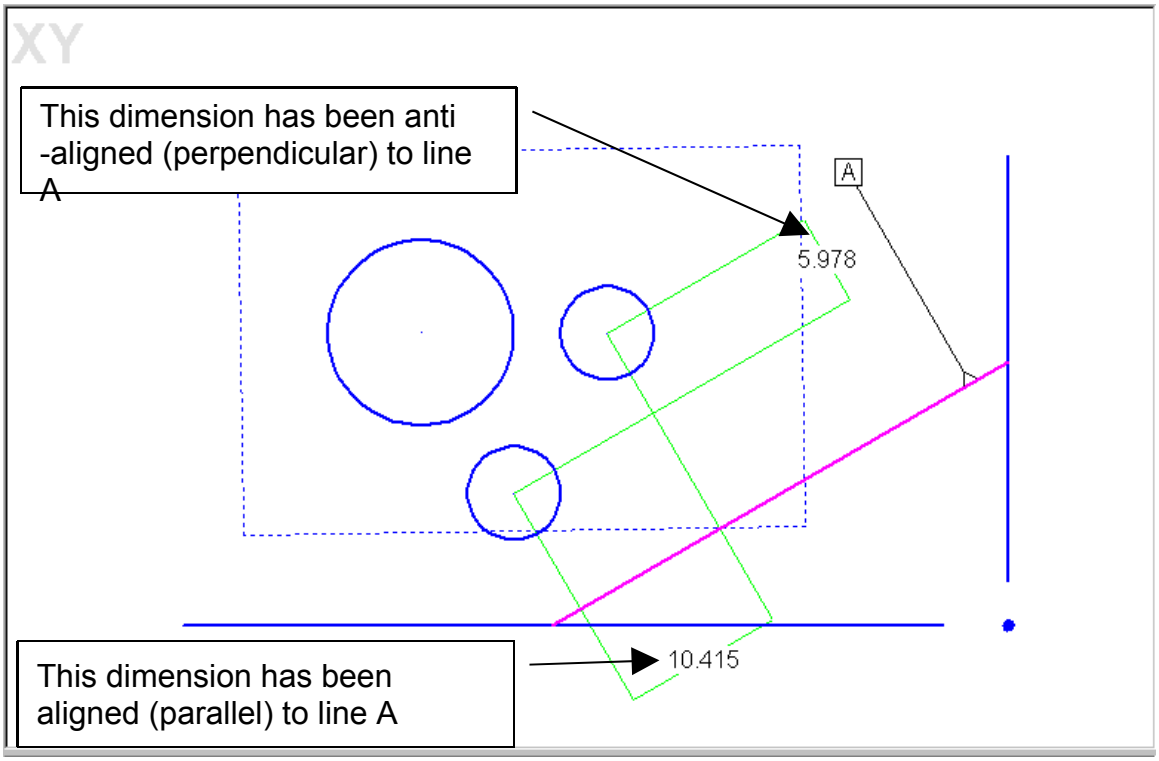


If however, the cone is selected as the second feature, the software will treat it as a point at its apex, and again produce a relevant dimension accordingly.


Aberlink 3D Software
File Edit Measurement Custom Utilities Stylus
Co-ordinate display, to zero a read out click on the panel, this has no effect on measurements or references.
90.009°
31.992
X +005.541
Y - 443.186
Z - 003.383
M-Grid
0

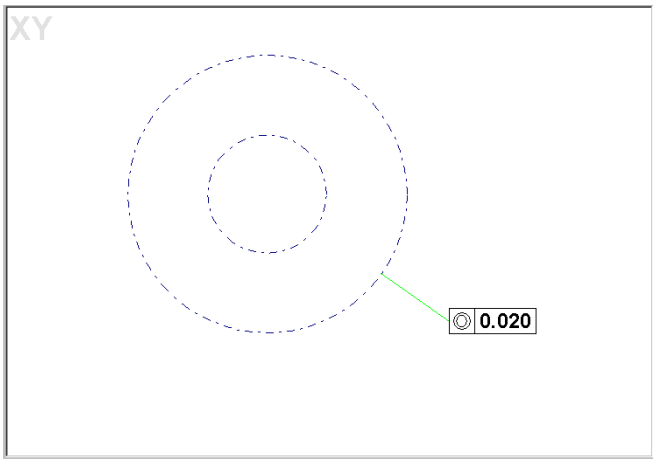
14.9 Aligned to Feature & Anti-aligned to Feature

A dimension which is aligned or ant-aligned (perpendicular) to a feature, will be displayed parallel or perpendicular to that feature, rather than the alignment that has been set for the inspection.



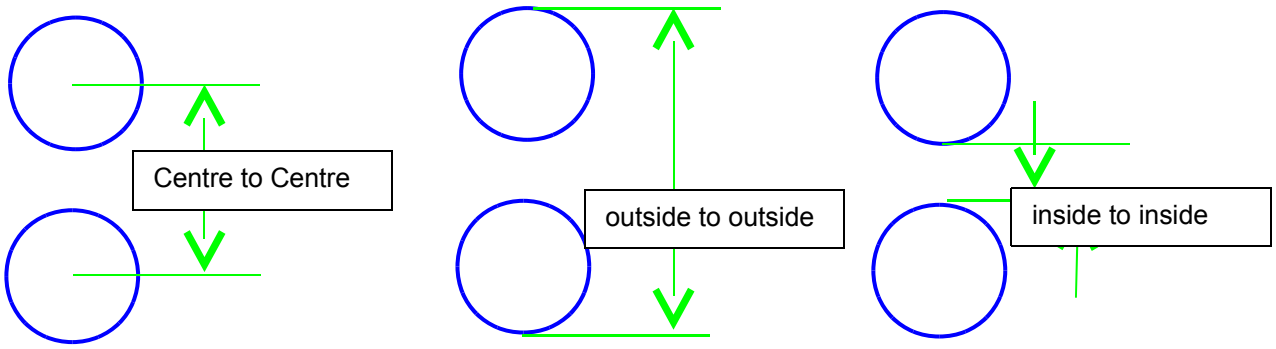
14.10 Leader option from 'Dimension type drop down menu'

If you need to know the concentricity between two selected circles or cylinders rather than the horizontal or vertical dimensions you can pick the 'Leader option' from the Dimension type drop down menu. This will give you a geometric box snowing the concentricity between the features.



14.11 Circle Max & Min Measure Option

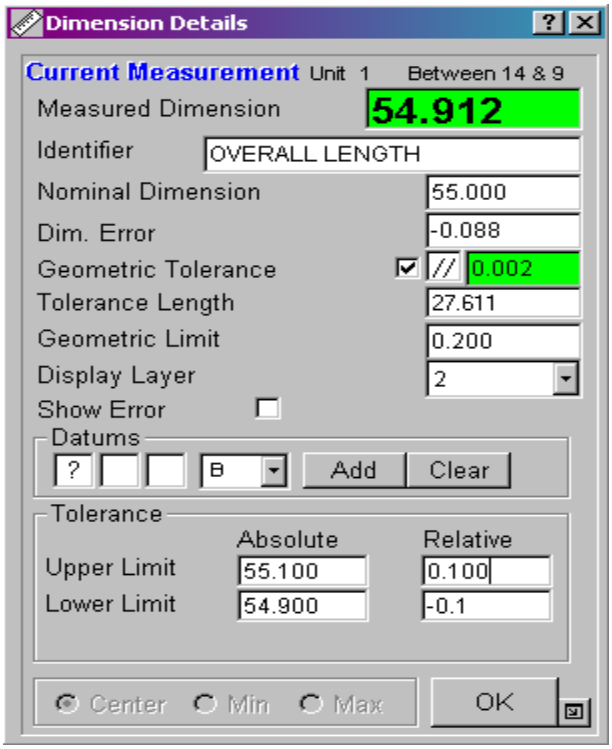
When measuring between circles (or a line and a circle) the software defaults to give the centre to centre distance. However it is also possible to dimension to either the inside or the outside of the circle(s):



Having called up the dimension between the centres of the circles, right click on the dimension to open the Dimension Details window. At the bottom of the window you will see Centre, Min and Max options to produce the dimension required.

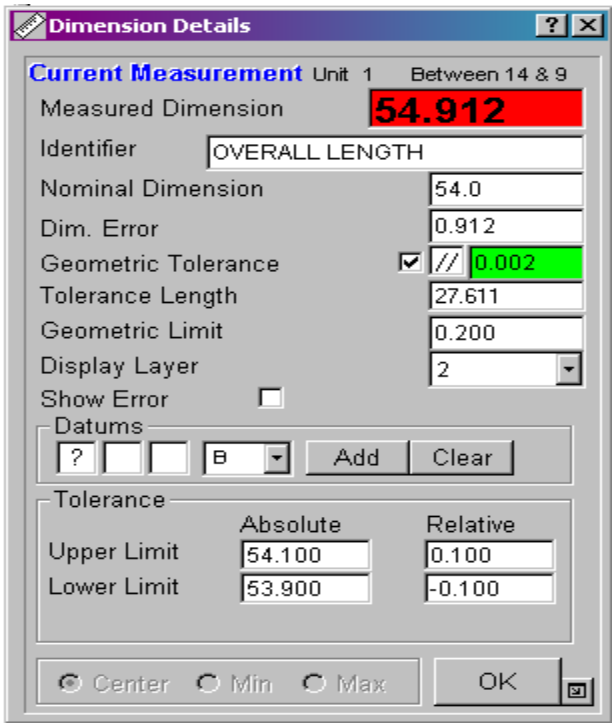
14.12 Setting Nominal Values and Tolerances for Dimensions

This is done within the Dimension Details window, which can be opened by right clicking on any dimension that has been called up.



Now the nominal dimension and tolerances can be set by simply typing over the default values that the software has selected.

Note also that if the measured dimension falls outside of the tolerance band set, then the background of the measured dimension value will become red.

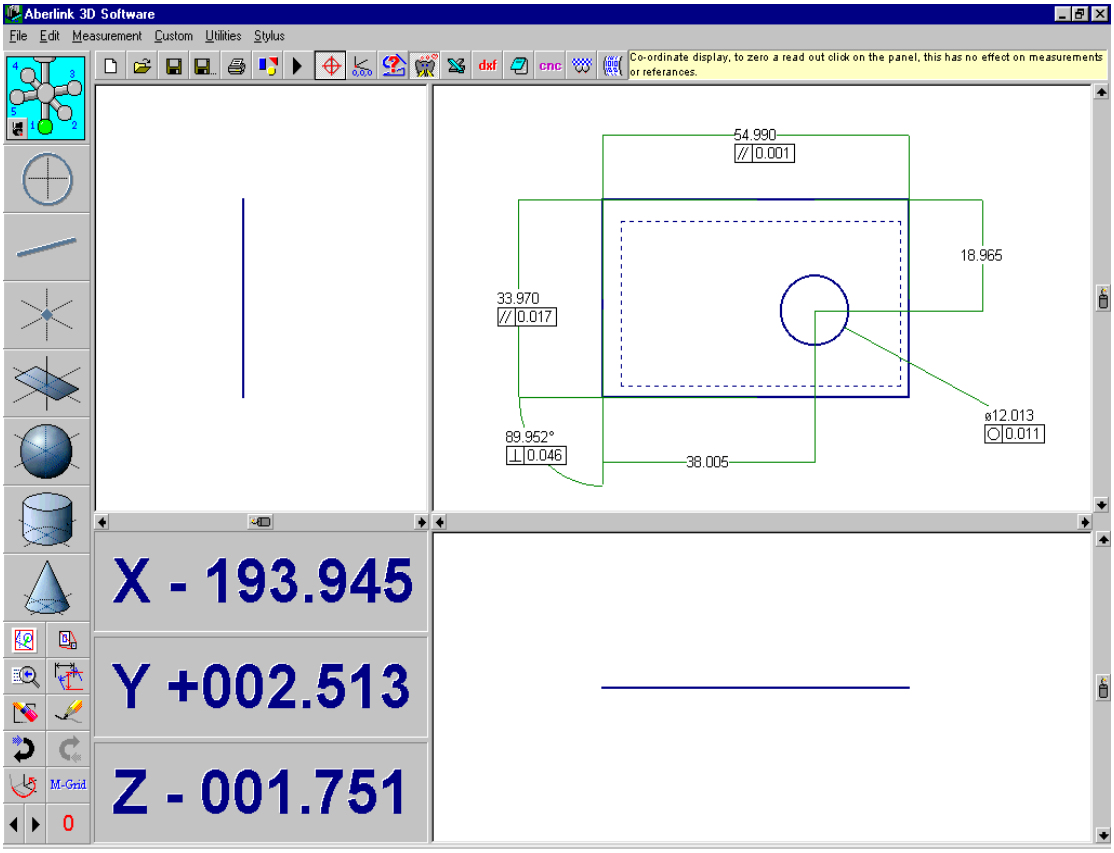


When you OK this window and return to the main screen, the dimension leader lines will also appear red on the graphical representation.

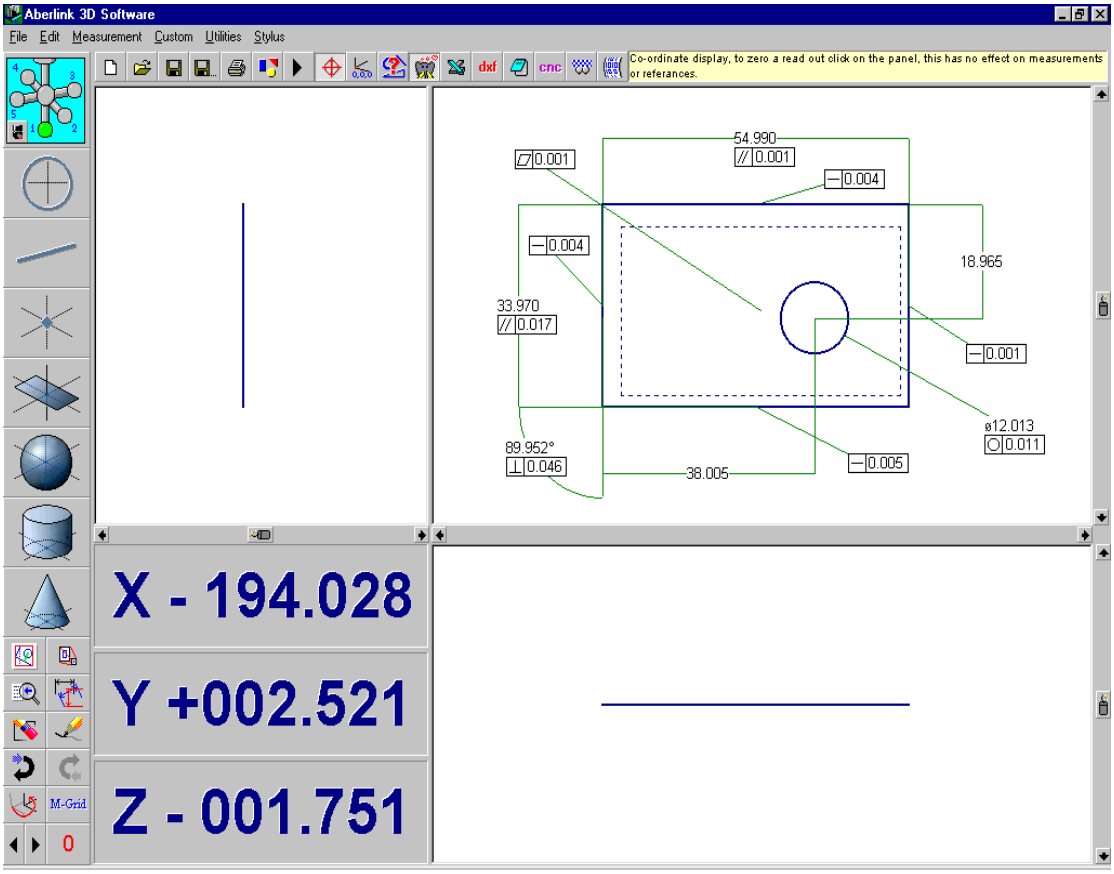
15.0 GEOMETRICAL TOLERANCES

To display on the screen all geometric tolerance values relevant to any dimensions called up click on the Geometric Tolerance button.

The geometric tolerance values that are relevant to the measurements on the screen will now be displayed.



Now to display the geometric tolerances that are relevant to any particular feature, (for instance flatness of a plane, straightness of a line) click twice on that feature and the outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and click for a third time. The geometric tolerance will now appear on the screen.



To turn all the Geometric Tolerances off simply click the Geometric Tolerance button again.

Individual tolerances can be switched off from within the Dimension Details window for that dimension. You will see a tick box adjacent to where the Geometric Tolerance is shown.

True Position for a circle can be shown by selecting the circle twice (as though calling up its diameter) but then right clicking and selecting either Cartesian or Polar position from the dimension type menu.

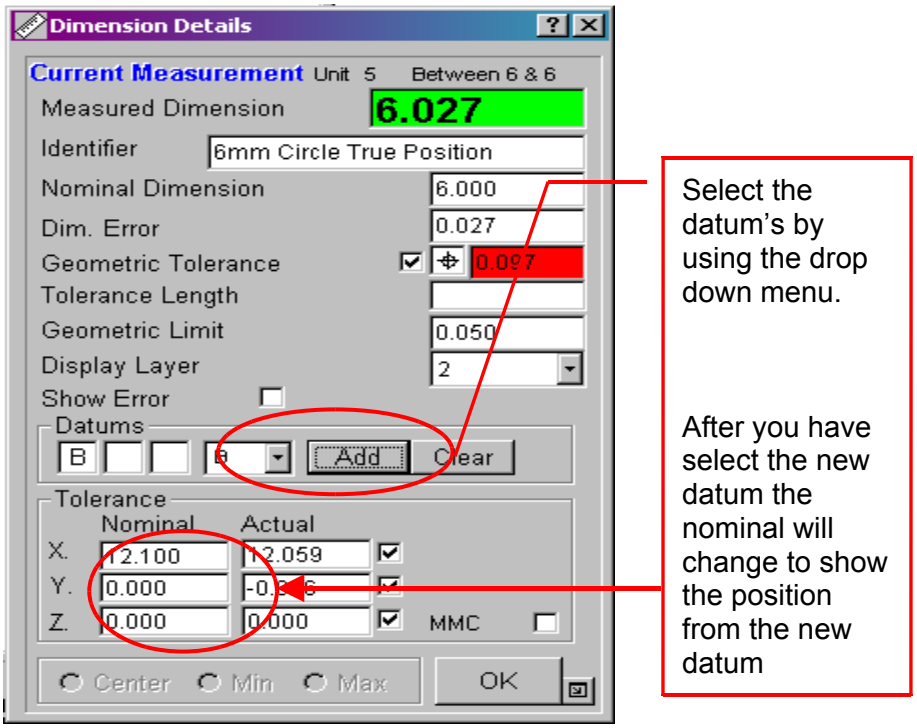
By default the position will be shown relative to the zero position that has been set for the inspection, however, it is also possible to show the True Position relative to other positions and alignments defined by datum features

To allocate a datum letter to a feature use the Datum pull down menu within the Measurement window for that feature. It is possible to display the datum letters in the graphic windows by clicking on a feature twice then right clicking and choosing datum from the dimension type menu.

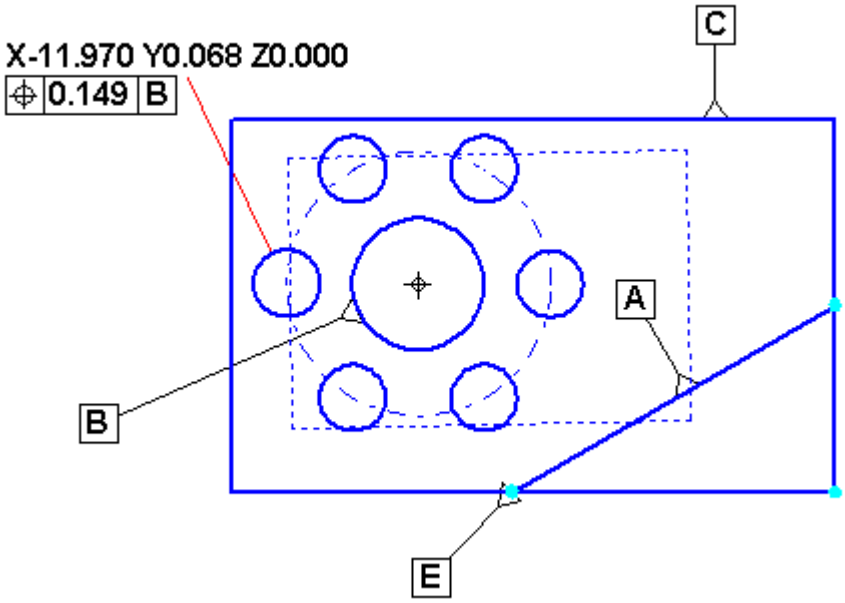
15.1 True Position relative to specified datum(s)

If the true position that you are measuring is relative to some other point on the component other than that which has been set as 0,0,0 then it is possible to show it relative to other specified datums.

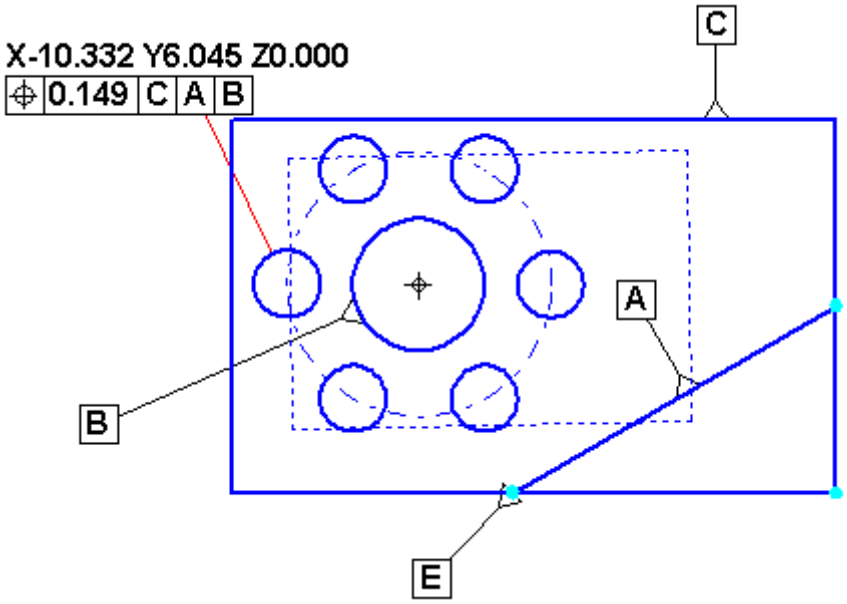
Note:- *to display the true position of a circle or a point from features other than the origin you must to allocate datum letters to those features.*



Select datum B and click on the 'Add' button. Also make sure that the nominal position of the hole is now relative to this datum. When you now return to the main screen the true position will now be displayed relative to datum B:



It is possible to select up to 3 datums relative to a true position:

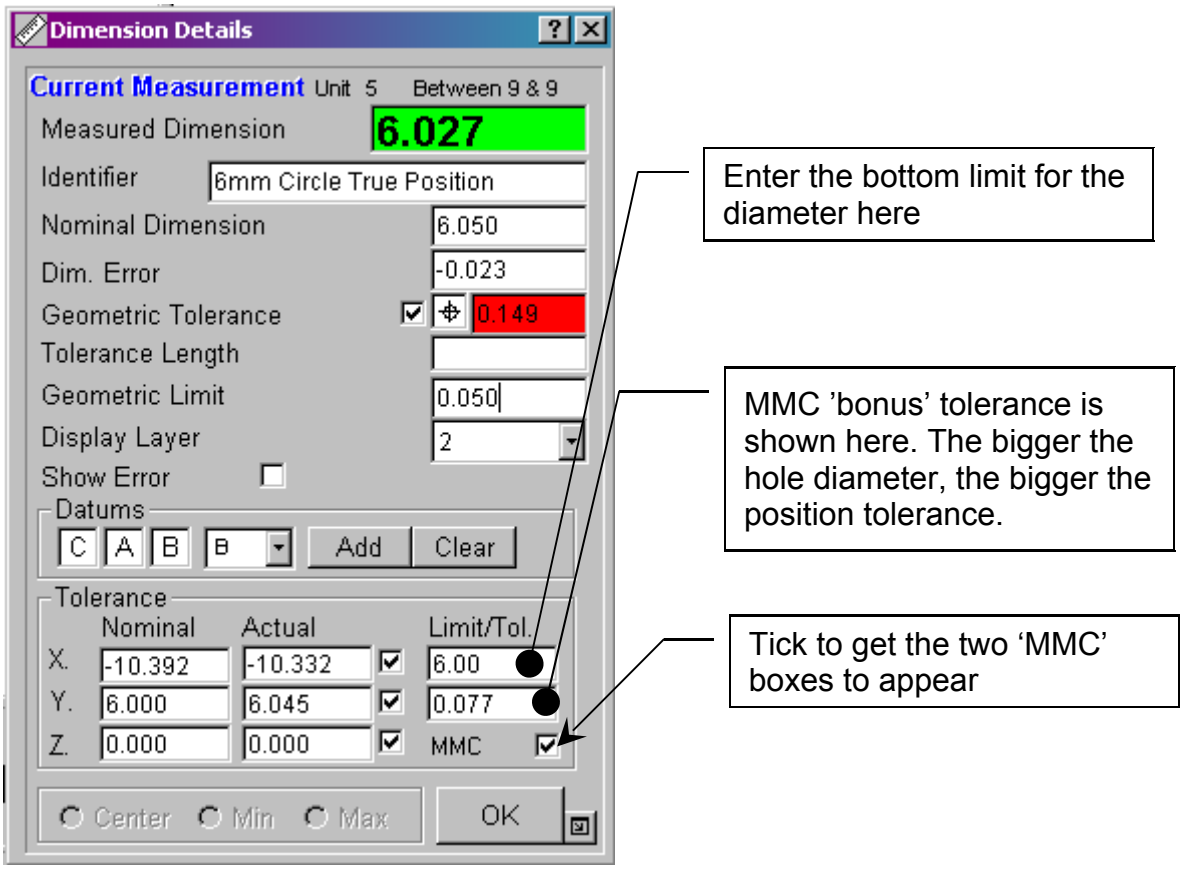


The position of the circle is at the bottom of the details section. If you change these nominal values the true position value will change accordingly. Note that these values are relative to the datum position on the component being measured.

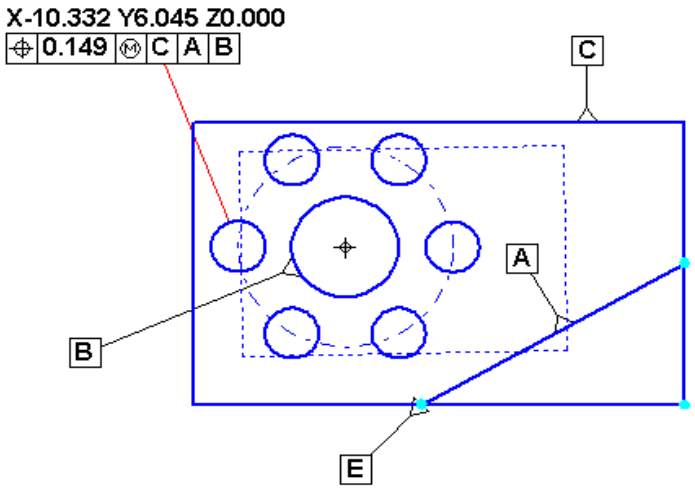
15.2 Maximum Material Condition

The maximum material condition for a hole can also be displayed if required. Again in the dimension details window tick the ‘MMC’ box. This will make two

further boxes appear. In the top box type the minimum allowable diameter for the hole. The lower box will then be automatically filled in with the ‘bonus’ tolerance due to the ‘MMC’:

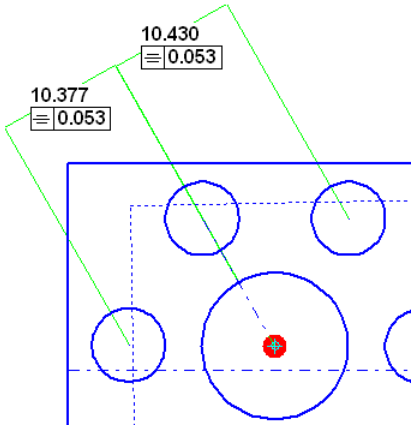

Dimension Details
Current Measurement Unit 5 Between 9 & 9
Measured Dimension 6.027
Identifier 6mm Circle True Position
Nominal Dimension 6.050
Dim. Error -0.023
Geometric Tolerance 0.149
Tolerance Length
Geometric Limit 0.050
Display Layer 2
Show Error
Datums
C A B B Add Clear
Tolerance
Nominal Actual Limit/Tol.
X. -10.392 -10.332 6.00
Y. 6.000 6.045 0.077
Z. 0.000 0.000 MMC
Center Min Max OK
Enter the bottom limit for the diameter here
MMC ’bonus’ tolerance is shown here. The bigger the hole diameter, the bigger the position tolerance.
Tick to get the two ‘MMC’ boxes to appear

When you return to the main screen the ‘MMC’ symbol will be present and the true position tolerance will be adjusted by the bonus tolerance:



15.3 Symmetry

If the geometric tolerance button is switch on, the symmetry between two similar dimensions will be displayed automatically underneath each dimension in the graphic window



16.0 GENERAL INFORMATION ABOUT MEASURED UNITS

16.1 Recalling a measured Unit

When you ‘OK or ‘Set Ref’ a measurement unit, and the measurement window disappears and the measured feature is displayed on the main screen, it is not too late to modify the unit by adding or deleting measurement points, or changing its reference status.

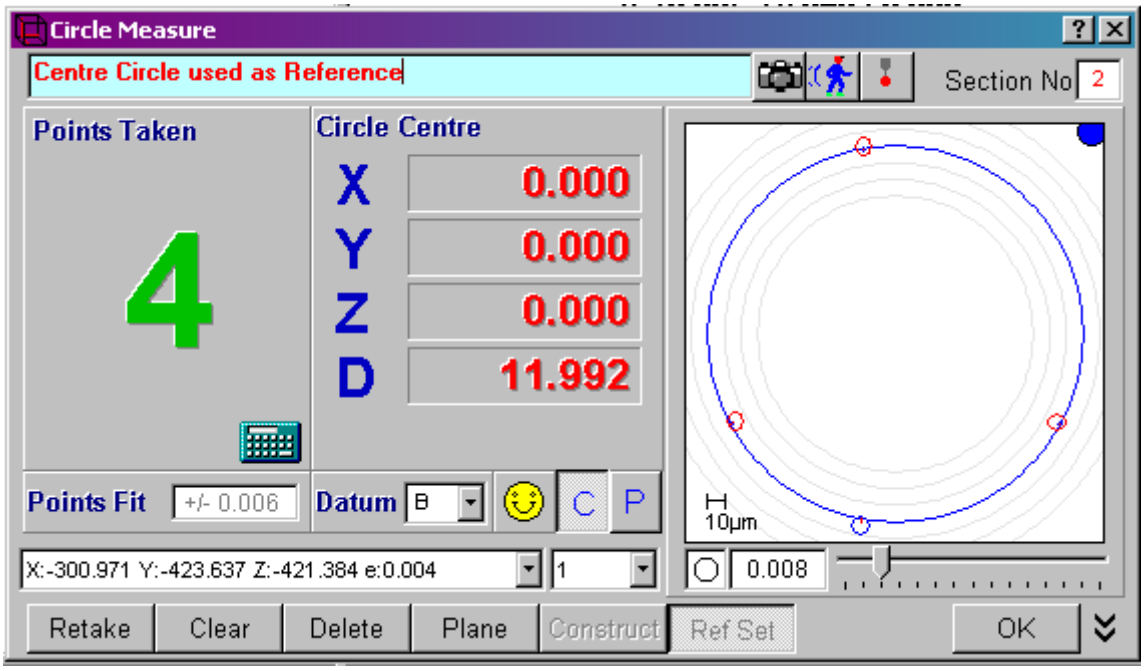
Each measured unit can be recalled by right mouse clicking on the graphical representation of the feature in the main screen.

Having recalled a measurement unit it can be modified in exactly the same way as when first being measured. Hence the ‘Retake’, ‘Clear’ and ‘Delete’ buttons are all functional, or alternatively extra measurement points may be taken on the feature. When you have completed the required modifications the ‘OK’ and ‘Set Ref’ plus ‘OK’buttons apply as usual.

16.2 Labelling Specific Measurement Units

The light blue bar at the top of every measurement unit is designed to allow text to be typed in it, in order to label each specific measurement unit. This may be useful if an inspection consists of similar or close features.

To insert text, simply click on the bar, and then enter the text via the keyboard.

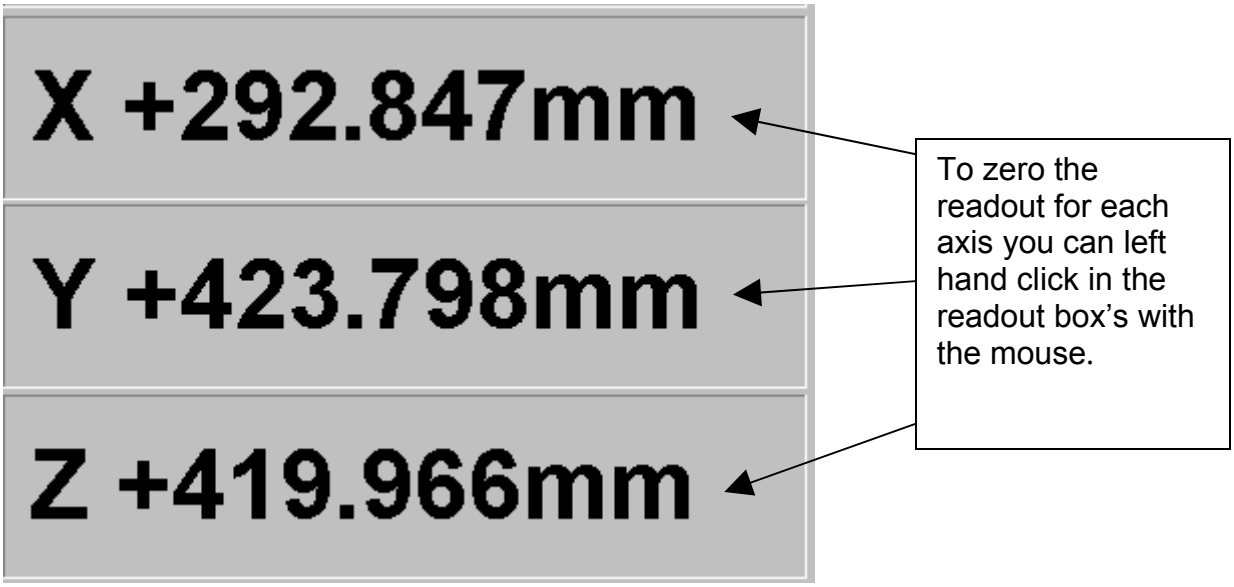


This may be particularly useful when using the Play function.

19.0 QUICK MEASUREMENTS USING THE SCALE READ OUTS

It is possible to take quick linear measurements on some features by simply touching the probe on two different positions on the component being measured and taking the readings from the scale readouts on the screen. These measurements however, should only be regarded as quick checks, as they cannot be saved or printed, and will only take into account single measurement points, with no stylus ball compensation.

This type of measurement is aided by the fact that the scale readouts can be set to zero by clicking on them with the mouse.


X +292.847mm
Y +423.798mm
Z +419.966mm
To zero the readout for each axis you can left hand click in the readout box's with the mouse.

Click on the readout for a second time to return to the absolute reading.

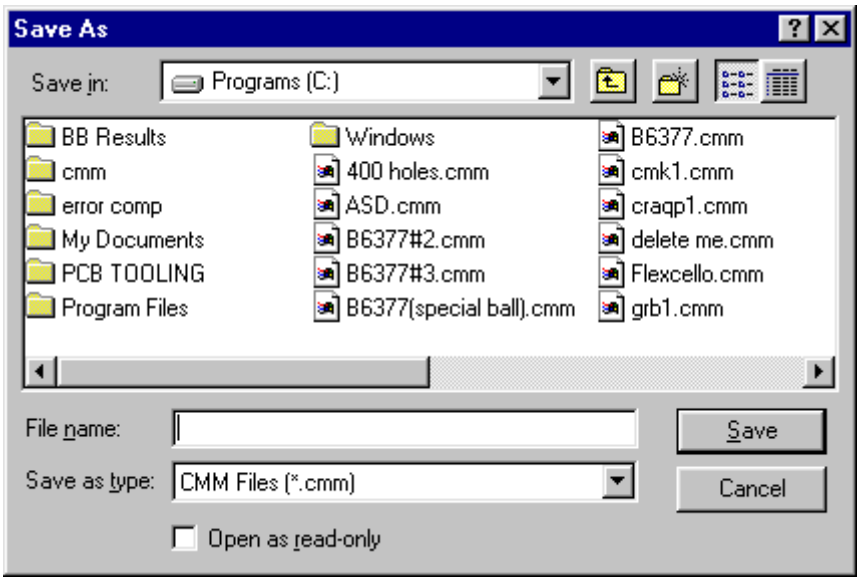
Note that right clicking on the readouts allows the possibility to display the readouts relative to the component reference position, or the machine's zero position, or alternatively to show an isometric view of the measured features.

18.0 SAVING MEASUREMENTS

When you have completed an inspection you can save the measurements either by clicking on the ‘Save File’ button or the ‘Save As’ button.

18.1 Save As

The Save As button should be used when you wish to save the inspection allocating it a new file name. When clicking on the Save As button the following window will appear.



Browse to where you want the inspection saved and give it a name.

18.2 Save File

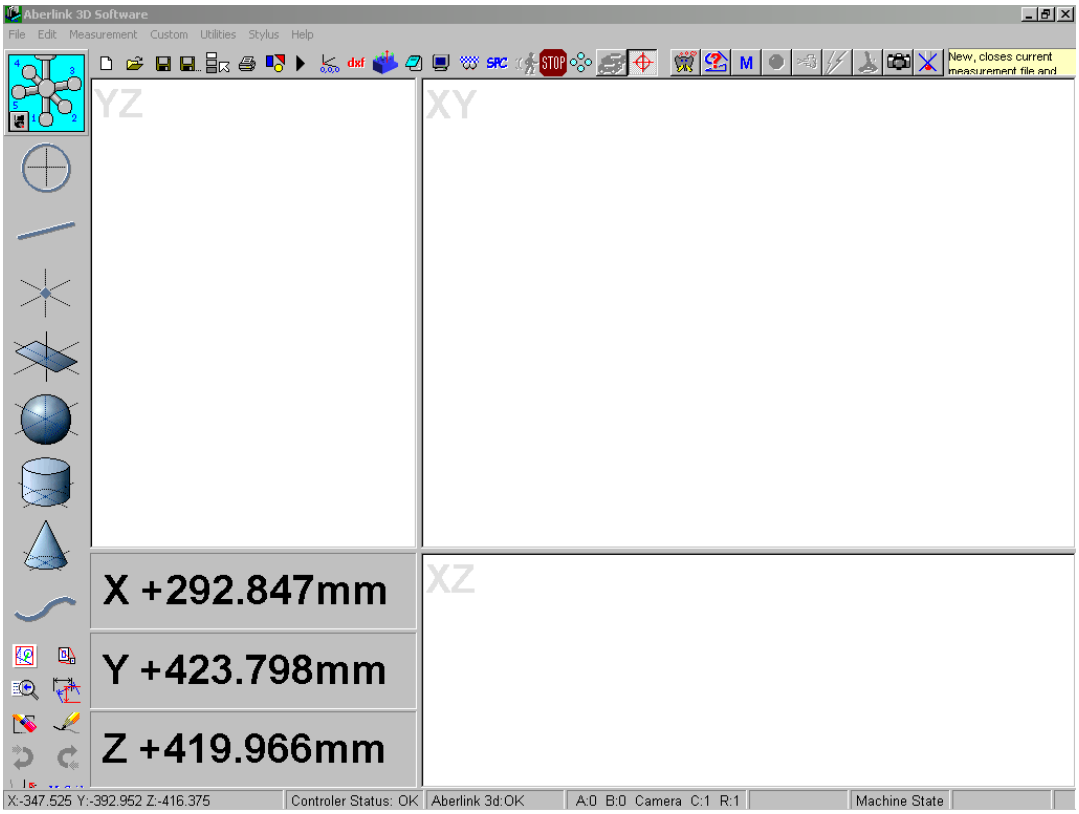
The Save File button should be used when the inspection has been previously saved, and you are happy to overwrite the saved file with the current inspection.

If you click on the Save File button and the inspection has not been previously saved, then you will be prompted to enter a file name, as for the Save As button.

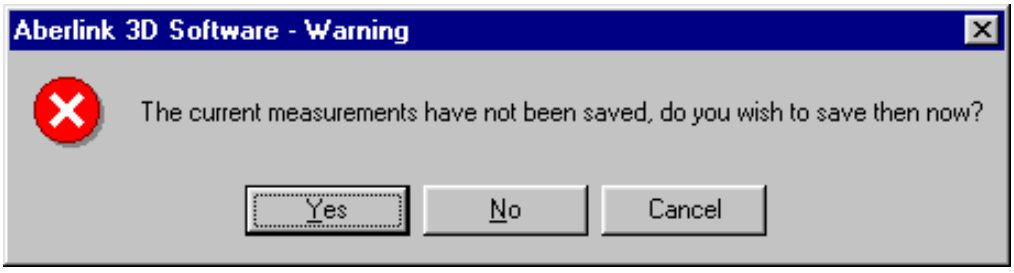
19.0 STARTING A NEW INSPECTION

If you need to start a new inspection at any time, click on the 'New Inspection' button.

If the previous inspection has already been saved as the last action, the graphics screens will go blank, and the software will be ready to start a new inspection.



If however, something has been modified in the last inspection, subsequent to the last time that the inspection was saved, then the following prompt will appear.

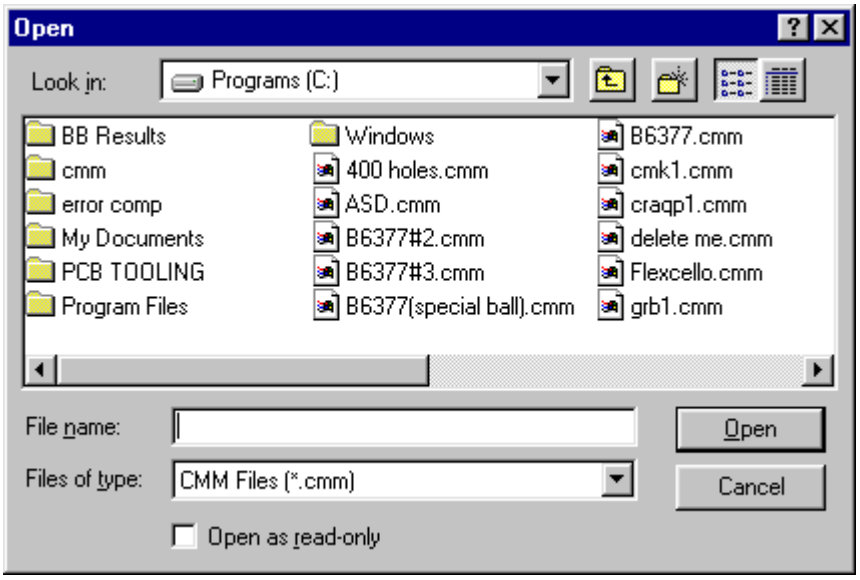


This gives you the opportunity to save the previous inspection in its final state before starting a new one.

20.0 OPENING A PREVIOSLY SAVED INSPECTION

If you wish to open an inspection that has previously been saved, click on the 'Open File' button.

When clicking on the Open File button the following window will appear.



Browse to where the inspection is saved and click on the Open button.

21.0 NAME AND SAVE FILE

This button allows the password protection of files.

When you click on this button the following window will appear:

If you type in a password, the inspection cannot be overwritten at any time in the future unless the password is again entered.

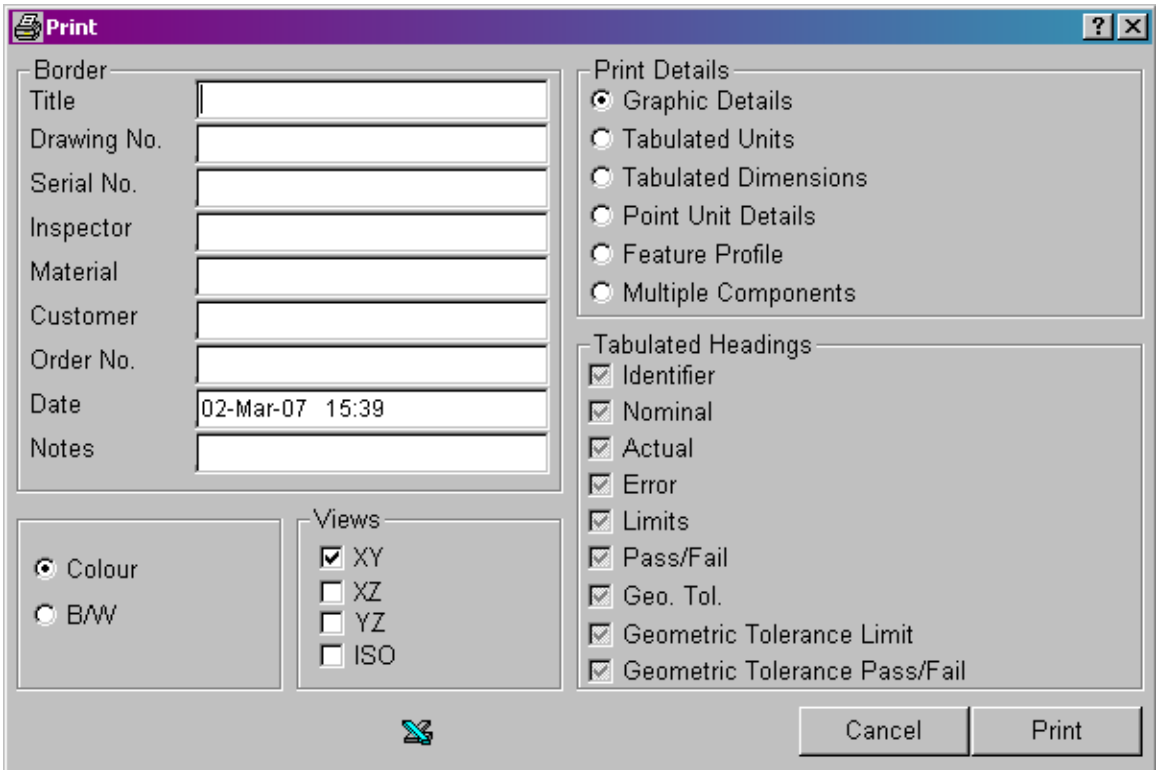
Note that it is possible to modify the inspection or add features to it, etc., however, these changes cannot be saved using the same name. Nevertheless, It is possible to save the inspection giving it a new name, using the ‘Save File As’ button.

When using a password be careful to remember the exact word, which will be case sensitive.

22.0 REPORTING RESULTS

Whenever you wish to produce a hard copy of inspection results the print function is used.

This will bring up the following window:



All inspection reports will be printed with a border around them, which can contain information about the part being inspected. This information can be entered in the border details frame. It is not essential to enter this information, and if not entered, the border will remain blank except for the time and date, which is entered by default. Your company name and address appears as entered in the Software Set Up.

Please note that the headings can also be customised by simply typing the required heading in this window.

There are six different types of report that can be printed:

- **Graphic Details,** prints results in the same form as the graphic display, i.e. an XY, XZ, YZ or isometric view of the Main Screen.

- **Tabulated Units,** prints the details of the each feature, similar to the information in the measure window of that feature.

- **Tabulated Dimensions,** prints all dimensions that are added to report from the current or previous inspections.

- **Point Positions,** prints the co-ordinates of points and the co-ordinates and size of circles.

- **Feature Profile,** prints the graphical representation of how well the individual points taken fit the theoretical shape i.e. the profile of a surface or the roundness of a circle.

- **Multiple Components,** by printing the dimensions in a line format you can get multiple parts on one sheet of paper.

You select the type of report that you require by clicking on the relevant option button.

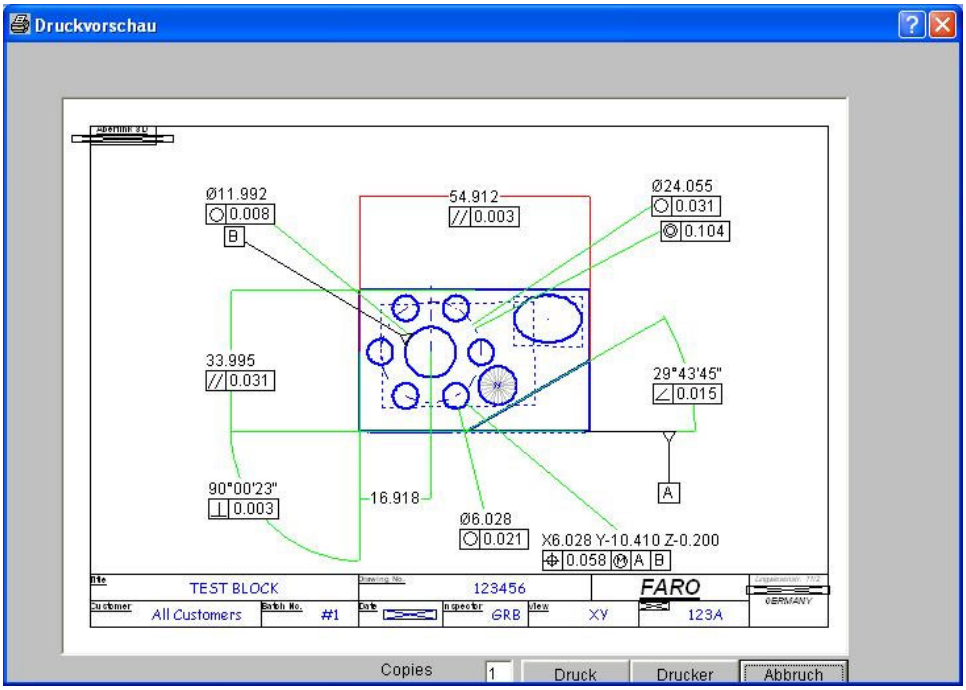
Choose whether you print out in colour or black and white.

When you click on the Print button for the first time you will get a Print Preview on the screen. If you are happy with this, click on the Print button again.

22.1 Printing Reports

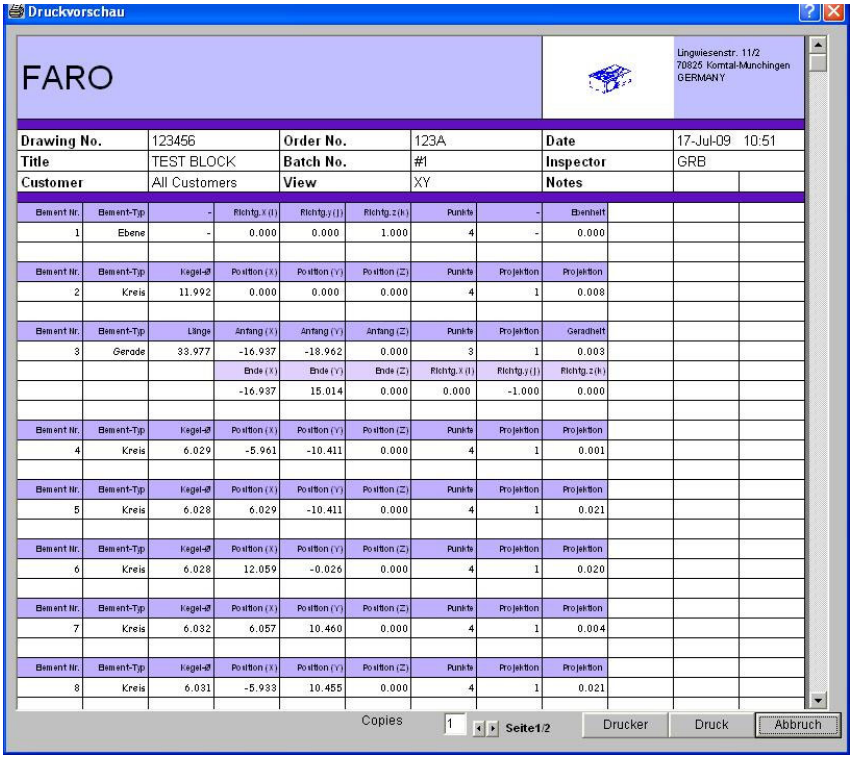
22.1.1 Graphic Details

This option prints out the Main Screen in a report format. If more than one view is selected the individual views are printed on separated sheets. The extent of the area that is printed is based on the current software view. If all necessary dimensions are not on the print preview then re-zoom the view in the software.



22.1.2 Tabulated Units

This routine will print the properties of the inspected measurement features eg. for a circle unit its centre co-ordinates and diameter will be printed or for a line unit the start and end co-ordinates will be printed, as well as its directional vector.

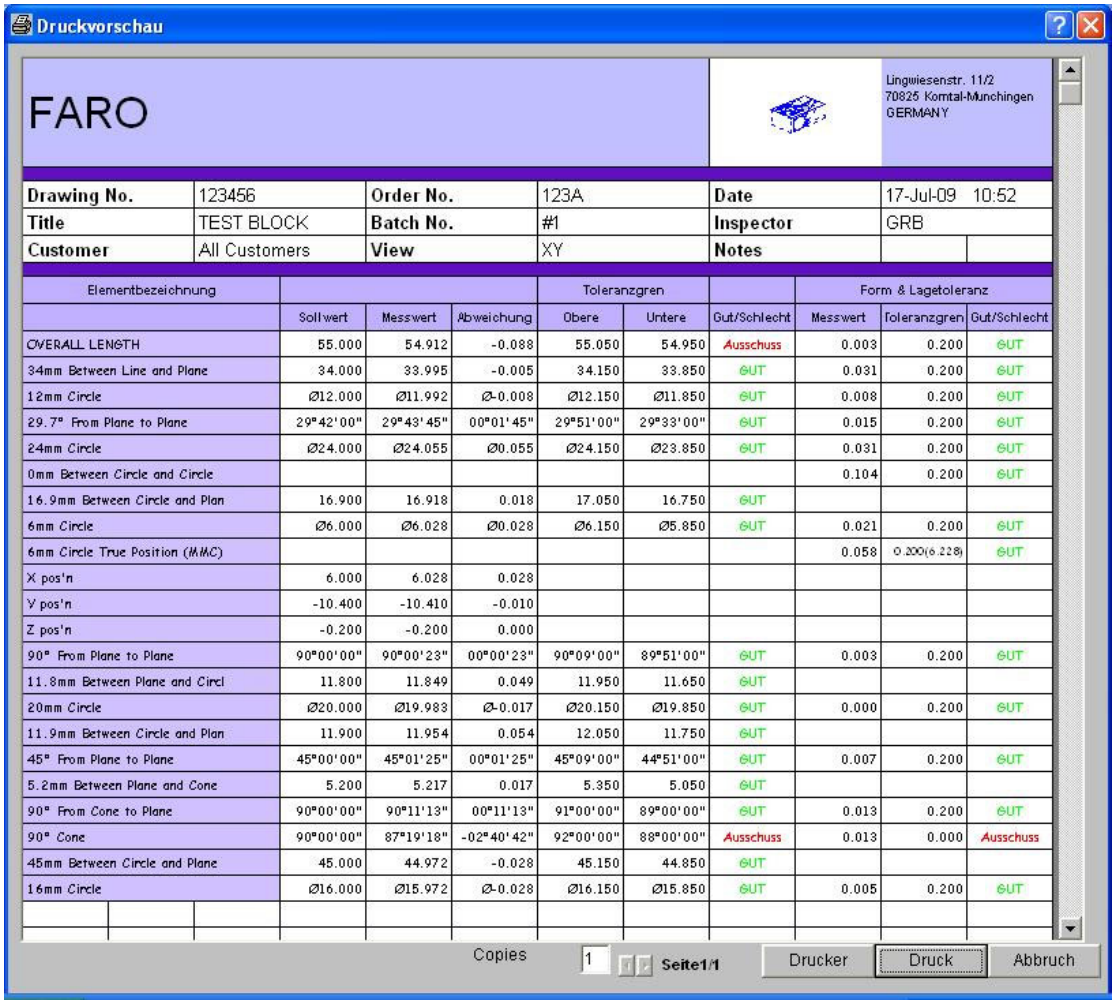

Druckvorschau
FARO
Lingwiesenstr. 11/2
70825 Korntal-Munchingen
GERMANY
Drawing No. 123456
Order No. 123A
Date 17-Jul-09 10:51
Title TEST BLOCK
Batch No. #1
Inspector GRB
Customer All Customers
View XY
Notes
Element Nr. Element-Typ - Richtg.x (i) Richtg.y (j) Richtg.z (k) Punkte - Ebenheit
1 Ebene - 0.000 0.000 1.000 4 - 0.000
Element Nr. Element-Typ Kegel-Ø Position (X) Position (Y) Position (Z) Punkte Projektion Projektion
2 Kreis 11.992 0.000 0.000 0.000 4 1 0.008
Element Nr. Element-Typ Länge Anfang (X) Anfang (Y) Anfang (Z) Punkte Projektion Geradheit
3 Gerade 33.977 -16.937 -18.962 0.000 3 1 0.003
Ende (X) Ende (Y) Ende (Z) Richtg.x (i) Richtg.y (j) Richtg.z (k)
-16.937 15.014 0.000 0.000 -1.000 0.000
Element Nr. Element-Typ Kegel-Ø Position (X) Position (Y) Position (Z) Punkte Projektion Projektion
4 Kreis 6.029 -5.961 -10.411 0.000 4 1 0.001
Element Nr. Element-Typ Kegel-Ø Position (X) Position (Y) Position (Z) Punkte Projektion Projektion
5 Kreis 6.028 6.029 -10.411 0.000 4 1 0.021
Element Nr. Element-Typ Kegel-Ø Position (X) Position (Y) Position (Z) Punkte Projektion Projektion
6 Kreis 6.028 12.059 -0.026 0.000 4 1 0.020
Element Nr. Element-Typ Kegel-Ø Position (X) Position (Y) Position (Z) Punkte Projektion Projektion
7 Kreis 6.032 6.057 10.460 0.000 4 1 0.004
Element Nr. Element-Typ Kegel-Ø Position (X) Position (Y) Position (Z) Punkte Projektion Projektion
8 Kreis 6.031 -5.933 10.455 0.000 4 1 0.021
Copies
1
Seite1/2
Drucker
Druck
Abbruch

22.1.3 Tabulated Dimensions

The dimensions that have been added to a part inspection can be printed in tabulated form. This is similar to a first article inspection report.

When printing Tabulated Dimension reports, it is possible to select what information is printed in the report using the check boxes in the Print window.

When you click on the 'Add to Report' button all the dimension in the current inspection will be added into the tabulated report.


Druckvorschau
FARO
Lingwiesenstr. 11/2
70825 Korntal-Munchingen
GERMANY
Drawing No. 123456 Order No. 123A Date 17-Jul-09 10:52
Title TEST BLOCK Batch No. #1 Inspector GRB
Customer All Customers View XY Notes
Copies 1 Seite1/1 Drucker Druck Abbruch

Elementbezeichnung	Sollwert	Messwert	Abweichung	Toleranzgren Obere	Toleranzgren Untere	Gut/Schlecht	Form & Lagetoleranz Messwert	Form & Lagetoleranz Toleranzgren	Gut/Schlecht
OVERALL LENGTH	55.000	54.912	-0.088	55.050	54.950	Ausschuss	0.003	0.200	GUT
34mm Between Line and Plane	34.000	33.995	-0.005	34.150	33.850	GUT	0.031	0.200	GUT
12mm Circle	Ø12.000	Ø11.992	Ø-0.008	Ø12.150	Ø11.850	GUT	0.008	0.200	GUT
29.7° From Plane to Plane	29°42'00"	29°43'45"	00°01'45"	29°51'00"	29°33'00"	GUT	0.015	0.200	GUT
24mm Circle	Ø24.000	Ø24.055	Ø0.055	Ø24.150	Ø23.850	GUT	0.031	0.200	GUT
0mm Between Circle and Circle							0.104	0.200	GUT
16.9mm Between Circle and Plan	16.900	16.918	0.018	17.050	16.750	GUT			
6mm Circle	Ø6.000	Ø6.028	Ø0.028	Ø6.150	Ø5.850	GUT	0.021	0.200	GUT
6mm Circle True Position (MMC)							0.058	0.200(0.228)	GUT
X pos'n	6.000	6.028	0.028						
Y pos'n	-10.400	-10.410	-0.010						
Z pos'n	-0.200	-0.200	0.000						
90° From Plane to Plane	90°00'00"	90°00'23"	00°00'23"	90°09'00"	89°51'00"	GUT	0.003	0.200	GUT
11.8mm Between Plane and Circl	11.800	11.849	0.049	11.950	11.650	GUT			
20mm Circle	Ø20.000	Ø19.983	Ø-0.017	Ø20.150	Ø19.850	GUT	0.000	0.200	GUT
11.9mm Between Circle and Plan	11.900	11.954	0.054	12.050	11.750	GUT			
45° From Plane to Plane	45°00'00"	45°01'25"	00°01'25"	45°09'00"	44°51'00"	GUT	0.007	0.200	GUT
5.2mm Between Plane and Cone	5.200	5.217	0.017	5.350	5.050	GUT			
90° From Cone to Plane	90°00'00"	90°11'13"	00°11'13"	91°00'00"	89°00'00"	GUT	0.013	0.200	GUT
90° Cone	90°00'00"	87°19'18"	-02°40'42"	92°00'00"	88°00'00"	Ausschuss	0.013	0.000	Ausschuss
45mm Between Circle and Plane	45.000	44.972	-0.028	45.150	44.850	GUT			
16mm Circle	Ø16.000	Ø15.972	Ø-0.028	Ø16.150	Ø15.850	GUT	0.005	0.200	GUT

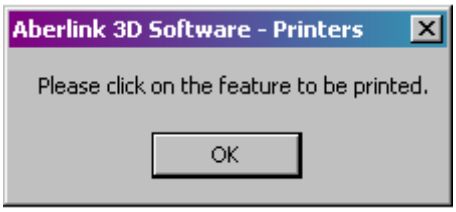
It is possible to add the dimensions from more than one inspection to a Tabulated Dimensions report, which is why the 'Clear Report' and 'Add to Report' buttons appear on the Print window. Clear Report removes all previous dimensions from the report, whilst the Add to Report button adds all the dimension in the current inspection into the report.

22.1.4 Point Positions

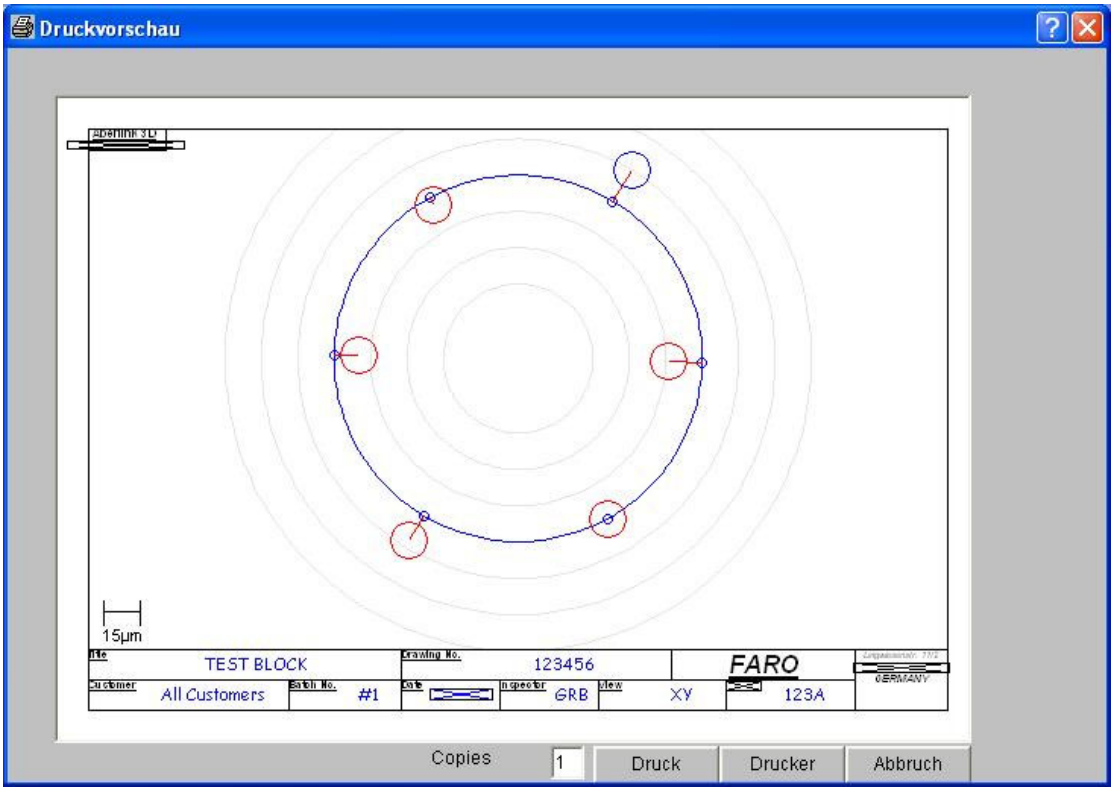
This print routine reports the X,Y,Z position and diameter of circles and points only. This is very useful if you are only interested in hole positions, for instance if inspecting a PCB or a drill jig.

22.1.5 Feature Profile

As a feature is inspected, the fit of the individual points to the theoretical shape is shown in the graphic window on the right side of the measurement window. This picture can be printed using the feature profile routine. When you click on Print button the software asks you to click on the feature you wish to print:



When you click on OK, the feature profile will be displayed as follows:



22.1.6 Multiple Components

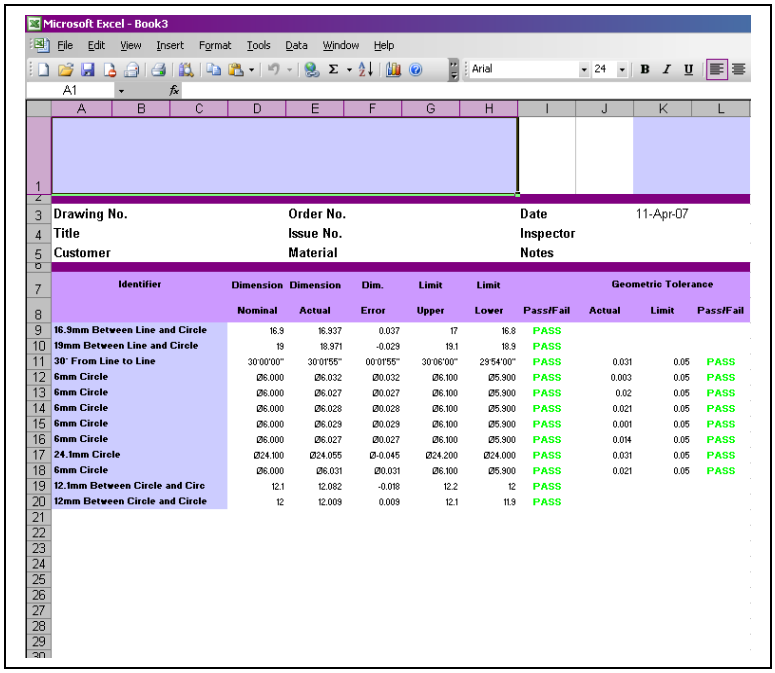
If when measuring a batch of components using the 'Play' function with 'SPC All Components' ticked on, the Aberlink 3D software will collect all the measured data for the entire batch. These values can be printed out in a single report using this option:

Note:- that any dimensions out of tolerance will be shown in red.

22.2 Exporting Reports to Excel

If Excel has been installed on the PC, then any tabulated report can be output an inspection to Execl by simply clicking on the Excel button within the Print window:

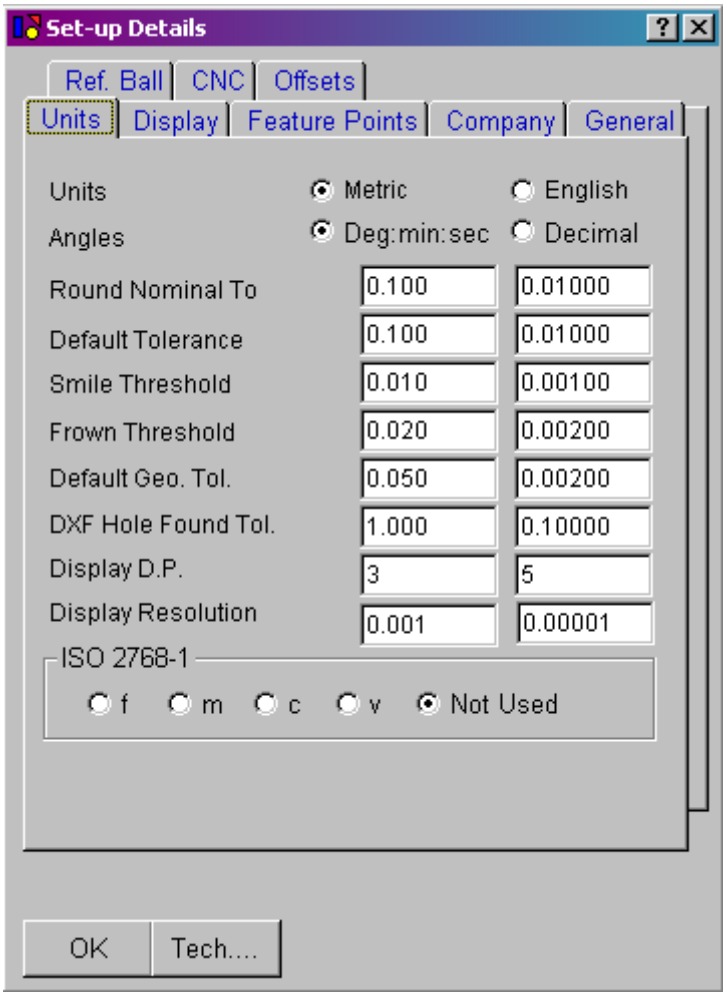
Excel will automatically be opened and the current inspection report will be loaded into a new worksheet:


Microsoft Excel - Book3
File Edit View Insert Format Tools Data Window Help
Arial 24
A1
Drawing No.
Order No.
Date 11-Apr-07
Title
Issue No.
Inspector
Customer
Material
Notes
Identifier
Dimension Nominal
Dimension Actual
Dim. Error
Limit Upper
Limit Lower
Pass/Fail
Geometric Tolerance Actual
Geometric Tolerance Limit
Geometric Tolerance Pass/Fail
16.9mm Between Line and Circle 16.9 16.937 0.037 17 16.8 PASS
19mm Between Line and Circle 19 18.971 -0.029 19.1 18.9 PASS
30° From Line to Line 30°00'00" 30°01'55" 00°01'55" 30°06'00" 29°54'00" PASS 0.031 0.05 PASS
6mm Circle Ø6.000 Ø6.032 Ø0.032 Ø6.100 Ø5.900 PASS 0.003 0.05 PASS
6mm Circle Ø6.000 Ø6.027 Ø0.027 Ø6.100 Ø5.900 PASS 0.02 0.05 PASS
6mm Circle Ø6.000 Ø6.028 Ø0.028 Ø6.100 Ø5.900 PASS 0.021 0.05 PASS
6mm Circle Ø6.000 Ø6.029 Ø0.029 Ø6.100 Ø5.900 PASS 0.001 0.05 PASS
6mm Circle Ø6.000 Ø6.027 Ø0.027 Ø6.100 Ø5.900 PASS 0.014 0.05 PASS
24.1mm Circle Ø24.100 Ø24.055 Ø-0.045 Ø24.200 Ø24.000 PASS 0.031 0.05 PASS
6mm Circle Ø6.000 Ø6.031 Ø0.031 Ø6.100 Ø5.900 PASS 0.021 0.05 PASS
12.1mm Between Circle and Circ 12.1 12.082 -0.018 12.2 12 PASS
12mm Between Circle and Circle 12 12.009 0.009 12.1 11.9 PASS

23.0 SOFTWARE SET UP

Many of the parameters used by the software may be set by the user to suit their particular requirements. In order to change any of these parameters click on the Set Up button:

This will bring up the Set Up window, as follows:



Clicking on the heading tabs will bring up each of the lists of user definable parameters. To change any of the parameters simply type in the new value over the existing.

23.1 Units

This is the information shown in the figure above. The 2 columns of boxes allow different values to be entered when working in metric values (mm) compared to when working in English values (inches):

23.1.1 Metric or English

Use the option buttons to select whether the results are given in millimetres or inches.

23.1.2 Display Angles as Deg, Min, Sec or as a decimal

Use the option buttons to select whether angles are reported as degrees, minutes and seconds or decimal angles. When changing between the two options the screen will need to be refreshed before showing the new angle information. The Redraw function will do this

23.1.3 Round Nominals To

When calling up a dimension the software will use the first measured value rounded to whatever value is set in this box. For example if the round nominal to value is set to 0.1 and a measured dimension is 19.965, then the software will use a default value for the nominal of 19.965 rounded to the nearest 0.1, ie. 20.0

23.1.4 Default Tolerance

Unless otherwise specified the software will apply +/-the value given as the tolerance for each dimension.

23.1.5 Smile / Frown Threshold

When measuring any type of feature there are 2 levels of warning that are given to let the operator know if a particular measurement point does not fit the form of the feature very well. The thresholds at which these warnings are given can be set in these boxes The Smiley Face feature will remain smiling providing that the error of the last measurement point does not exceed the threshold value set in the Smile Threshold box.

If the error of the last measurement point falls between the smile threshold and the frown threshold the smiley face feature will have an open mouth (frown). If the last measurement point error exceeds the frown threshold then the Smiley Face feature will become unhappy and an audible warning will be given.

23.1.6 Default Geo. Tol.

This value sets the default value for all Geometric Tolerances.

23.1.7 DXF Hole Found Tolerance

It is possible to import nominal hole positions for a component in a DXF file format. When doing this the software will intelligently pair the measured results to the positions defined in the DXF file. In this case the software will use the tolerance set in this box to determine whether a measured hole is paired with inputted values.

23.1.8 Display D.P.

Use this box to modify the number of decimal places shown on the readouts, measured dimensions and in all inspection reports.

23.1.9 Display Resolution

All readouts and measured dimensions will be rounded to this value, irrespective of the number of decimal places displayed. For example, if a measured dimension of 29.987 is shown to 3 decimal places with a display resolution of 0.1, the result will be displayed as 30.000. It is therefore important to match the display resolution to the number of decimal places used.

23.1.10 The ISO 2768 Tolerance standard

You can use the ISO 2768 tolerance standard to tolerance your parts by clicking on the buttons marked f, m, c & v (fine, medium, coarse and very course). The software will then apply the tolerances according to the standard and shown in the tables below.

Table 1 - Permissible deviations for linear dimensions except for broken edges (external radii and chamfer heights)

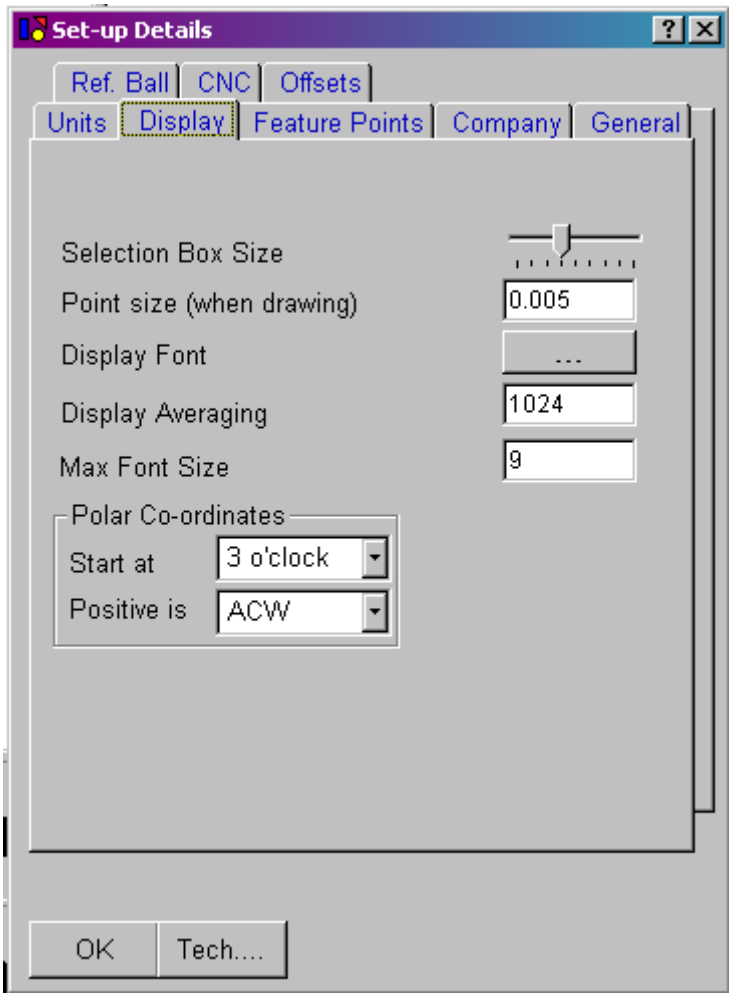
Tolerance class		Permissible deviations for basic size range							
Designation	Description	from 0,5* up to 3	over 3 up to 6	over 6 up to 30	over 30 up to 120	over 120 up to 400	over 400 up to 1000	over 1000 up to 2000	over 2000 up to 4000
f	fine	± 0,05	± 0,05	± 0,1	± 0,15	± 0,2	± 0,3	± 0,5	--
m	medium	± 0,1	± 0,1	± 0,2	± 0,3	± 0,5	± 0,8	± 1,2	± 2
c	coarse	± 0,2	± 0,3	± 0,5	± 0,8	± 1,2	± 2	± 3	± 4
v	very coarse	--	± 0,5	± 1	± 1,5	± 2,5	± 4	± 6	± 8
* For nominal size below 0,5 mm, the deviation shall be indicated adjacent to the relevant nominal size(s).									

Table 3 – Permissible deviations for angular dimensions

Tolerance class		Permissible deviations for ranges of lengths in millimetres of shorter side of the angle concerned				
Designation	Description	up to 10	over 10 up to 50	over 50 up to 120	over 120 up to 400	over 400
f	fine	± 1°	± 0,5°	± 0,333°	± 0,166°	± 0,083°
m	medium					
c	coarse	± 1,5°	± 1°	± 0,5°	± 0,25°	± 0,166°
v	very coarse	± 3°	± 2°	± 1°	± 0,5°	± 0,333°

Indication on drawing – example (medium): ISO 2768-m alternatively SS-ISO 2768-1 m

23.2 Display



23.2.1 Selection Box Size

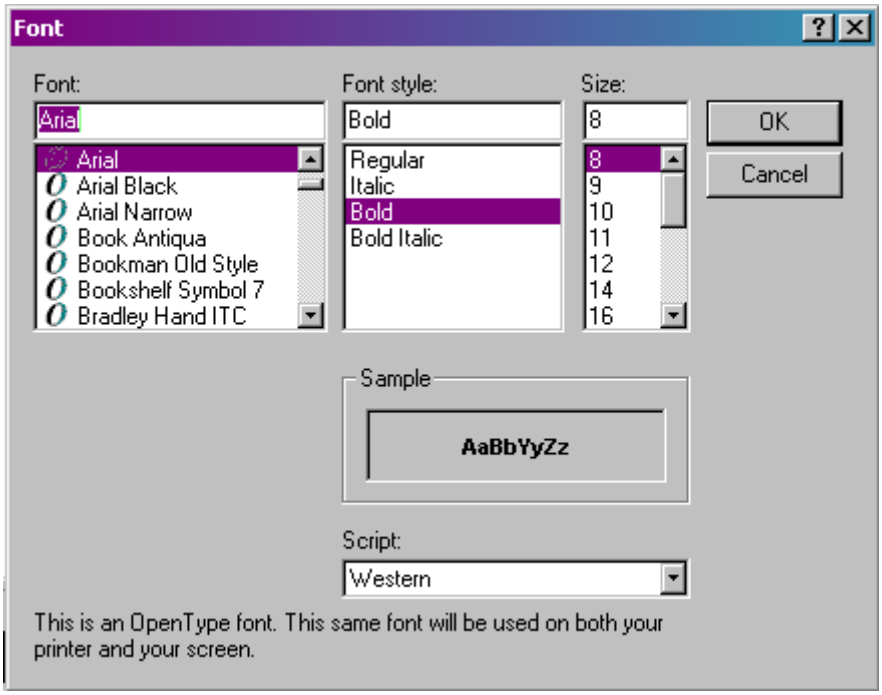
The slider bar can be used to adjust the size of the selection box that appears in the graphical areas of the Main Screen.

23.2.2 Point Size (when drawing)

When measuring or constructing points, they appear in the Main Screen as blue filled circles. Modifying the value in this box will alter the size of these points.

23.2.3 Display Font

The font, style and size of the results can be selected by clicking on the grey box. This will bring up the following window:



The font, style and size of the printed results may now be selected from the lists available.

23.2.4 Max Font Size

If the font size is bigger than 9 the letters will be too big to fit the buttons or drop down menus

23.2.5 Display Averaging

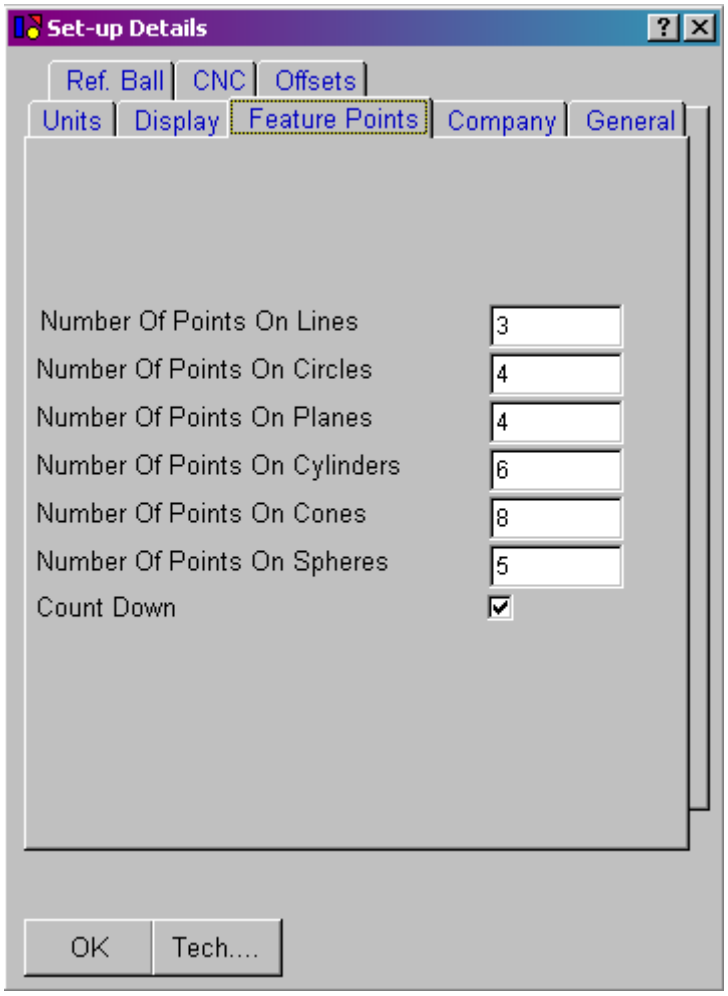
The software will take this number of position readings from the scales (max 1024) before displaying the average. By applying averaging the scales will appear smoother.

23.2.6 Polar Co-ordinates

BS 308 defines the angle of a polar co-ordinate as 3 o'clock on a clock face equating to zero degrees, and anti-clockwise defining a positive angle (hence 12 o'clock = 90 degrees and 6 o'clock = 270 degrees etc.)

If you wish to work to an alternative standard you can use these boxes to define zero degrees and which direction defines a positive angle.

23.3 Feature Points



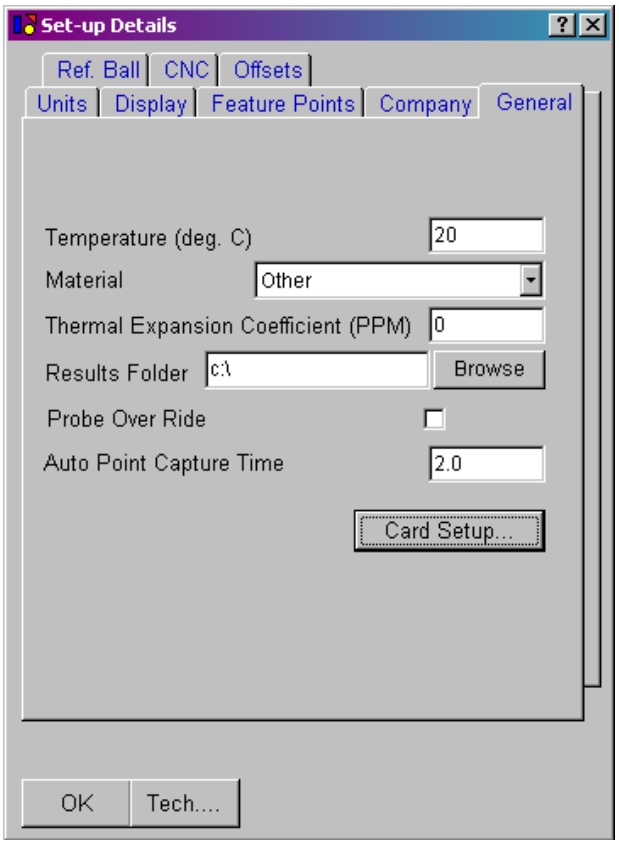
During normal operation, features are inspected by the operator taking an unspecified amount of measurement points, and the software will count these points upwards from zero. It is up to the operator to decide when they have taken sufficient points.

If however, the Count Down box in this window is ticked, then the measurement windows will start to count down from the specified number of points defined in the boxes above.

23.4 Company

The name and address shown in this window will appear on all printed inspection reports.

23.5 General



23.5.1 Temperature (deg C)

Temperature compensation is not available on a Faro arm or Gage. Please leave the temperature set at 20°C.

23.5.2 Material

Temperature compensation is not available on a Faro arm or Gage. Please leave the material set at 'Other'.

23.5.3 Thermal Expansion coefficient (PPM)

Temperature compensation is not available on a Faro arm or Gage. Please leave the Thermal Expansion Coefficient set at 0.

23.5.4 Results Folder

The path shown in this box will be the default path called up by the software when saving results using the Save File and Save As buttons.

23.5.5 Probe Override

Ticking this box will have the effect of ignoring the signal from the probe. This may be useful during faultfinding etc. when the probe may need to be bypassed.

This box would also need to be ticked if a solid probe were ever used in the place of the touch trigger probe.

23.5.6 Auto Point Capture Time

If you are using a solid probe on a manual machine to measure profiles by dragging the probe across the surface it is possible to capture points at set intervals over time.
You enter the time between points here.

23.6 Ref Ball Tab

Not relevant when used with a Faro Arm

23.7 CNC Tab

Not relevant when used with a Faro Arm

23.8 Offsets Tab

Not relevant when used with a Faro Arm

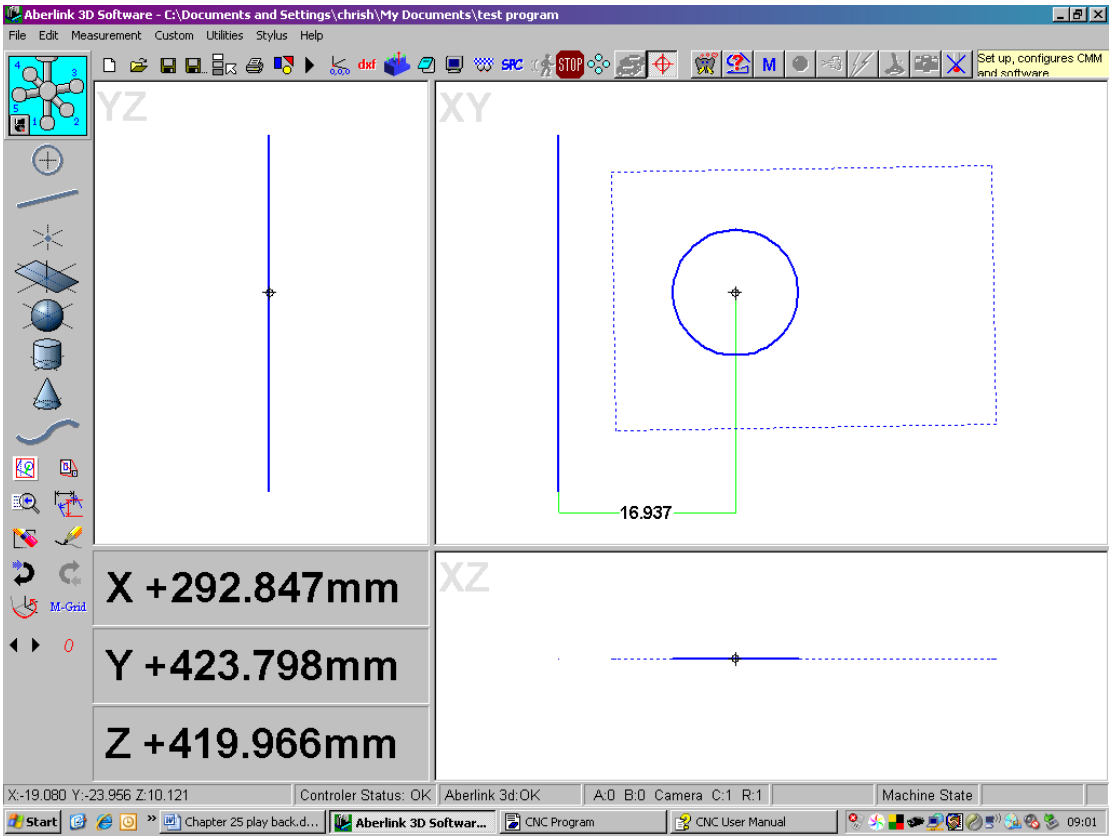
24.0 THE PLAY FUNCTION

Every time that you inspect a component, you are automatically creating a program to measure subsequent parts of the same type.

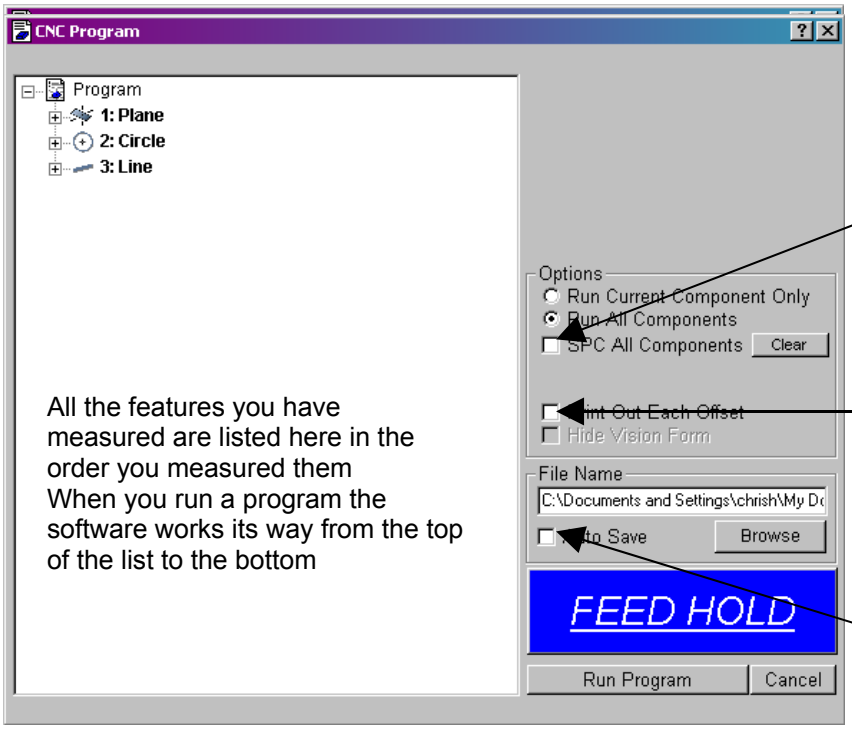
To run the program click on the ‘Play’ button

The play function will now take you through the inspection of the part in the same order as previously measured, by highlighting in green each of the features previously measured in turn.

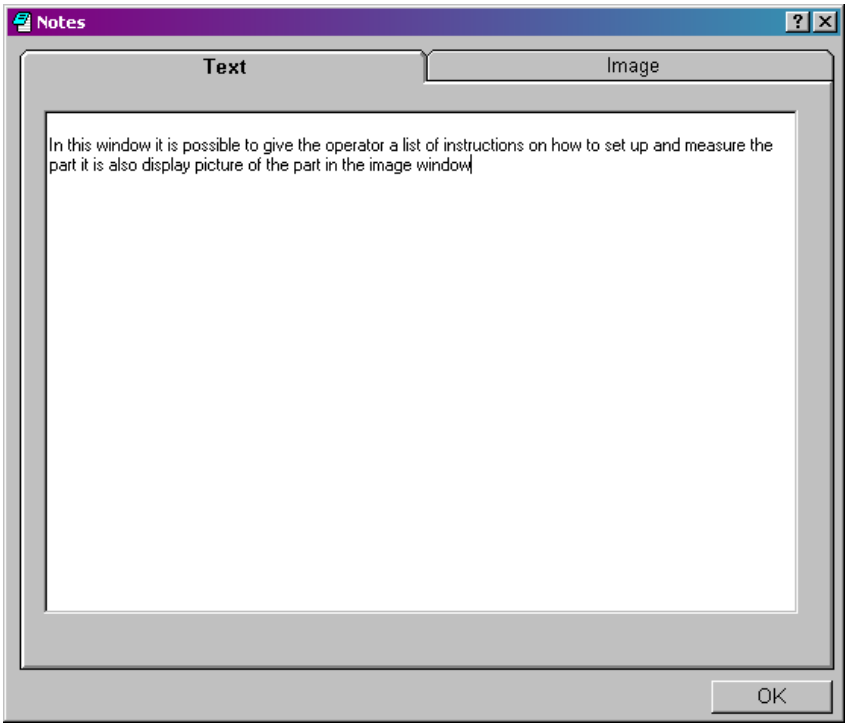
For instance, consider an inspection that consisted of a plane, a line and a circle measured in that order, with a single dimension giving the distance between the line and the circle.



Now to measure a similar component in the same manner, click on the Play button, this will open the program window shown below:


CNC Program
Program
1: Plane
2: Circle
3: Line
All the features you have measured are listed here in the order you measured them
When you run a program the software works its way from the top of the list to the bottom
Options
Run Current Component Only
Run All Components
SPC All Components
Clear
Hide Vision Form
File Name
Browse
FEED HOLD
Run Program
Cancel
If you click this Box the results are save automatically to be used with SPC.
If you click this Box the results are printed out automatically after each measurement.
If you click this Box the results are save automatically on the computer after each measurement.

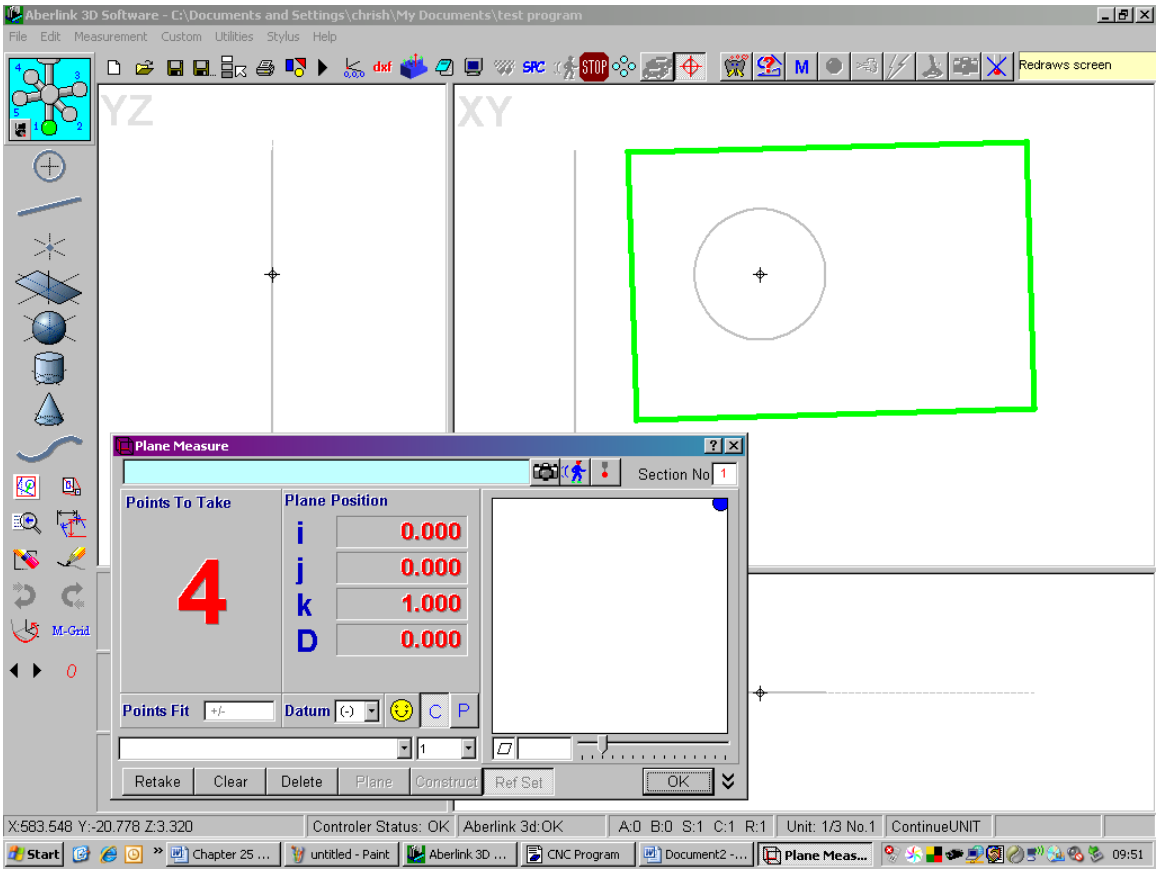
To run the program press the 'Run Program' button this will minimize the program window and open the 'Program Notes window' as shown below:


Notes
Text
Image
In this window it is possible to give the operator a list of instructions on how to set up and measure the part it is also display picture of the part in the image window
OK

In this window it is possible save notes or images that may me useful for the inspection.

Click ‘OK’ and the program will continue:

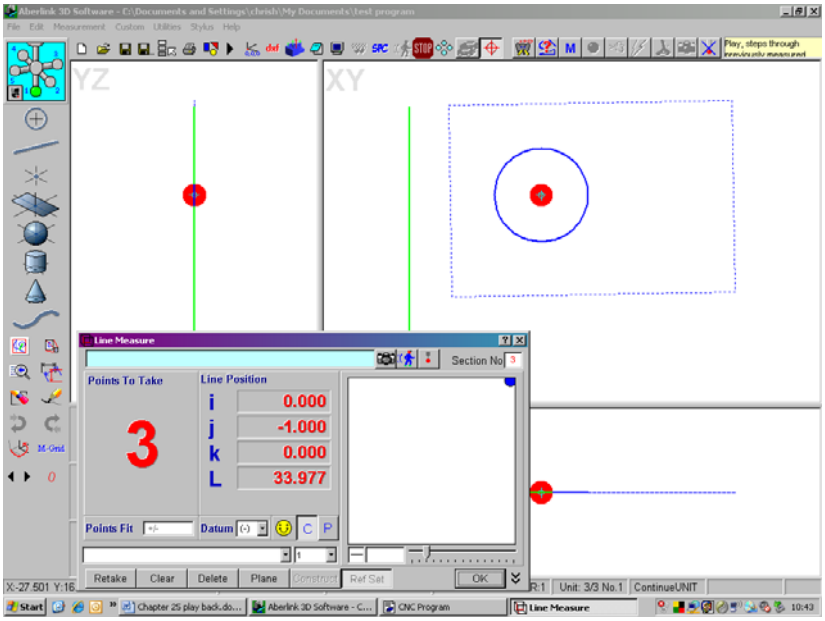
The first feature originally measured, the Plane, will become highlighted in green, and the Plane Measure window will automatically open showing the required number of points (same as when previously measured). Also the dimensions that were called up in the program will disappear.



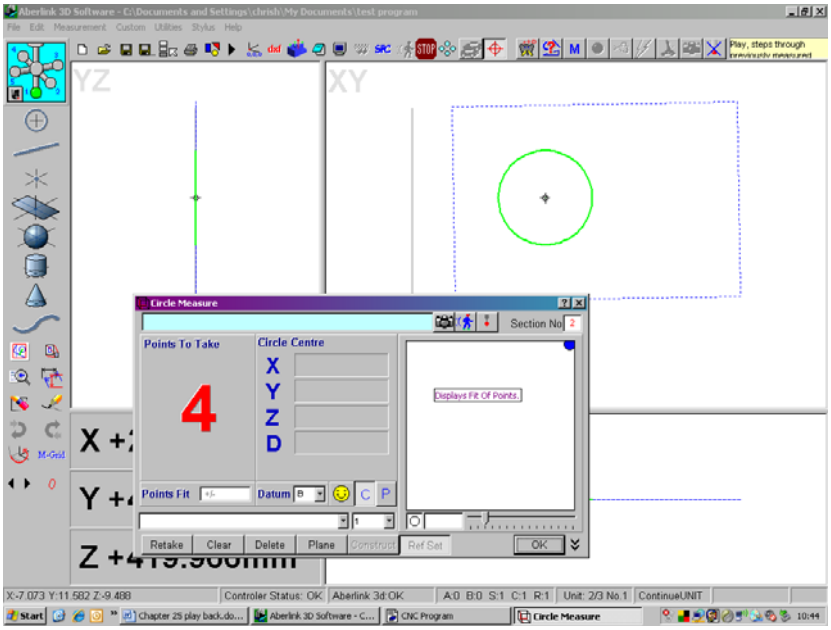
Note – that when using the Play function it may be useful to enter a description of each feature in the light blue box at the top of the measurement window as another means of identifying the correct feature.

Now measure 4 points on the plane on the second component. As you take the points the ‘Points to Take’ window will count down. When you take the last point the measurement window will automatically close, the new plane will be drawn in the Main Screen, and the next measurement window will open.

Also Note - At any time before you taking the last point you can retake or clear points. The last point can only be modified by going back into that unit at the end of the program.



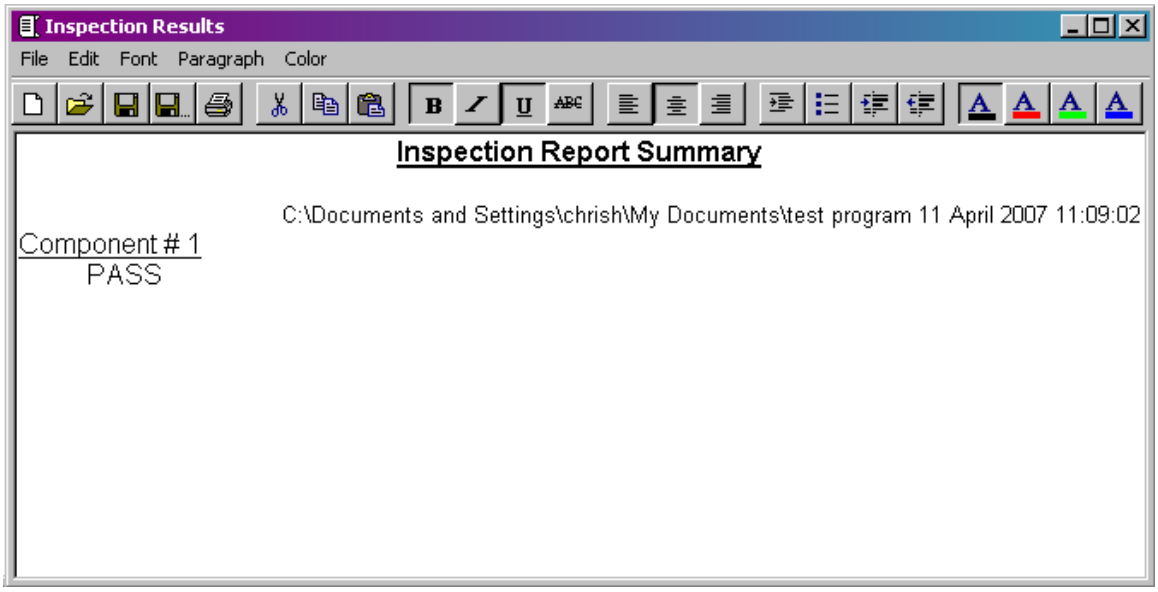
The line will be highlighted green to show which feature needs to be measured. Repeat the measurement of the line. After 3 points the Line Measurement window will automatically close, the new line will be drawn in, the next feature will be highlighted in green and its measurement window will automatically open:



Repeat the process for the circle measurement.

As features are re-measured, any dimensions that have been called up between them will automatically re-appear in the Main Screen. At the end of

the program a summary of any dimensions that are out of limits will be displayed:



To repeat the inspection again for a third component, you must click on the Play button again.

The Play function makes multiple inspections as quick as it can be on a manual measuring device. Measurement windows are automatically opened, and automatically closed when the required number of points has been taken. Dimensions are automatically re-created. Constructed Features are automatically re-created. Dimensions that are out of tolerance can be seen at a glance – this is a very powerful tool.

24.1 More information on the Program Tree

All the measurement units are shown in the program tree on the left hand side of the window. The reference features will be shown in bold.

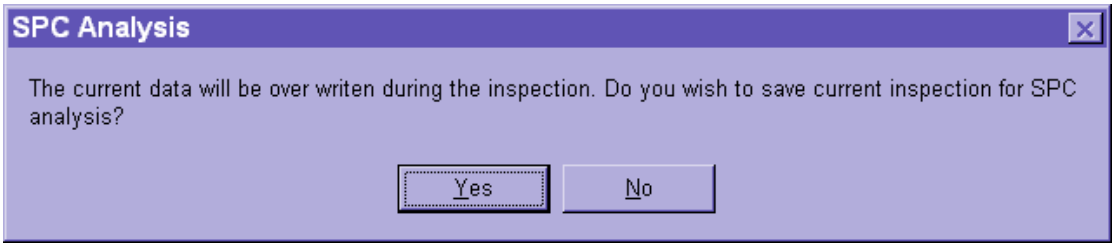
More information about the units can be obtained by clicking on the plus sign adjacent to it. It is possible to look at the template for the unit.

You can modify the order of the program by clicking on a unit within the tree and dragging it to a new position. Care however, must be taken not to infringe the basic rules of the Aberlink 3D measuring software such as dragging a circle or a line in front of the Plane into which they are projected.

24.2 Collecting SPC Batch Information

Another useful option when using the Play function is the ‘SPC All Components’ option. The Aberlink 3D software will collect batch information as well as calculate some simple SPC information and plot both a histogram and a scatter graph for any given dimension.

In order to collect this information when running a batch of components you need to tick the ‘SPC All Components’ box on the Program Window.
Now when you click on ‘Run Program’ you will get the following message box:


SPC Analysis
The current data will be over writen during the inspection. Do you wish to save current inspection for SPC analysis?
Yes
No

If you wish to add the previous inspection to the batch information click on ‘Yes’, otherwise click ‘No’.

The batch information can be displayed within the Dimension Details window for each dimension. See the chapter specifically regarding SPC information.

25.0 THE STORE POINTS FUNCTION

The Store Points function will save measured points to the memory, if no Measurement window is open.

This button is latching. Click on it again to turn it off.

If there is a point or points stored in the memory, these can be fitted to any geometric feature simply by opening a measurement window of the appropriate type.

Please note that when this feature is switched on, every point taken (when there is not a measurement window open) will be stored in the memory, even when taken by accident, and will be fitted to the next feature opened. Therefore unwanted points will have to be retaken when the measurement window is opened.

26.0 THE FEATURE PREDICT FUNCTION

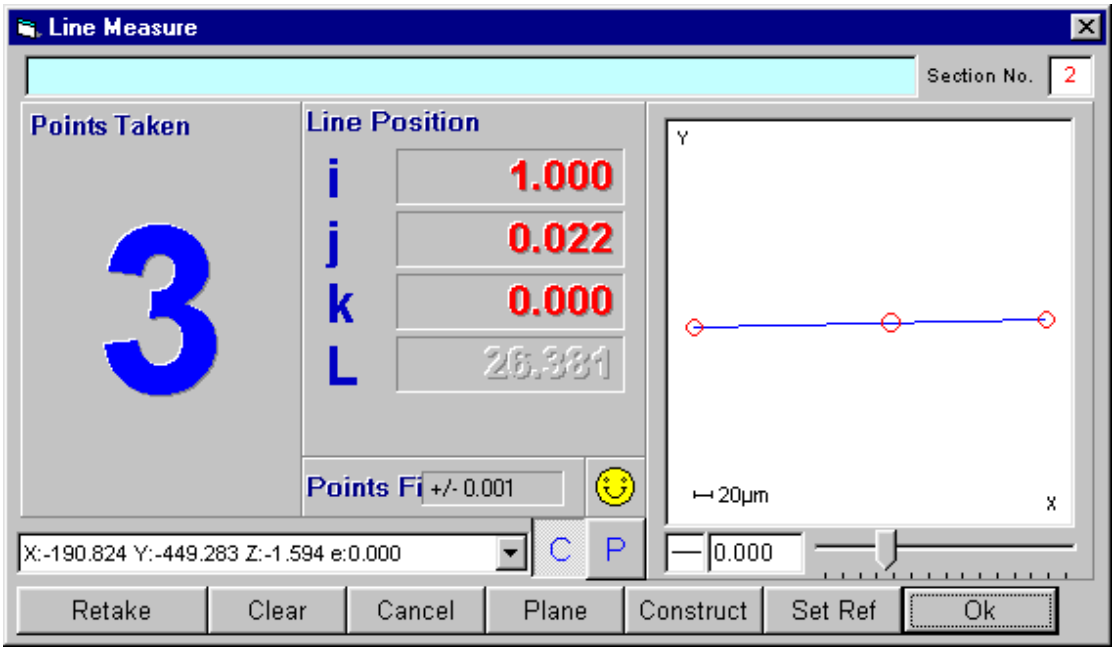
The Feature Predict function is a time saving function, aimed at cutting down the amount that the user has to interface with the PC. In this mode the software will predict the feature required to fit the measurement points taken. This function will work for 2 dimensional features, such as lines, circles and planes, but cannot be used for either one-dimensional points or 3 dimensional features such as cylinders, cones or spheres.

To put the software into this mode, click on the Feature Predict button.

This button is a latching type, which when on, will become highlighted. It will also automatically bring on the Store Points function, which will also become highlighted. To switch the Feature Predict function off again you must click on the button for a second time, and the button will return to its normal grey colour. Note that the Store Points function will also have to be turned off separately.

With the Feature Predict function switched on, you can immediately start to take the measurement points on a feature that you are measuring, without clicking on the measurement button first.

If the first 3 measurement points taken can be fitted to a line, a Line Measurement window will automatically appear on the screen displaying the points taken.



If however the points taken do not fit a line, a fourth measurement point must be taken. After this point the software will automatically call up a Circle Measure window or a Plane Measure window, depending on what type of feature it is able to fit the points to.

If the software has not called up any measurement window after 4 points it is best to click on the required measurement button to tell the software what shape to fit the measured points to.

You can continue to take measurement points for any unit, as desired, after the measurement window has appeared. The window will be updated accordingly. When you are ready to measure the next feature, there is no need to click on 'OK', simply start to take measurement points on the next feature. As soon as the software cannot fit a measurement point to the existing measurement shape, it will give a bad point noise, to warn that it is starting a new measurement, and the initial measurement window will close, with the previous feature now being displayed on the main screen. Continue taking measurement points on the second feature until the new measurement window appears after 3 or 4 points.

You can now continue in the same way until all the desired features have been measured. On the final feature you must click on 'OK' or 'Set Ref' to remove the final measurement window and return to the main screen.

> ***Note*** *– When you start to measure a new feature in this mode, and the previous measurement window closes, lines and planes will be set as a reference if no like feature has previously been referenced, or else it will be the equivalent of clicking OK.*

If at any stage whilst using the Feature Predict function you accidentally probe on the wrong point, you will need to interface with the computer to erase it.

If the measurement window for the feature you are measuring is already on the screen, the erroneous measurement point can be erased as normal.

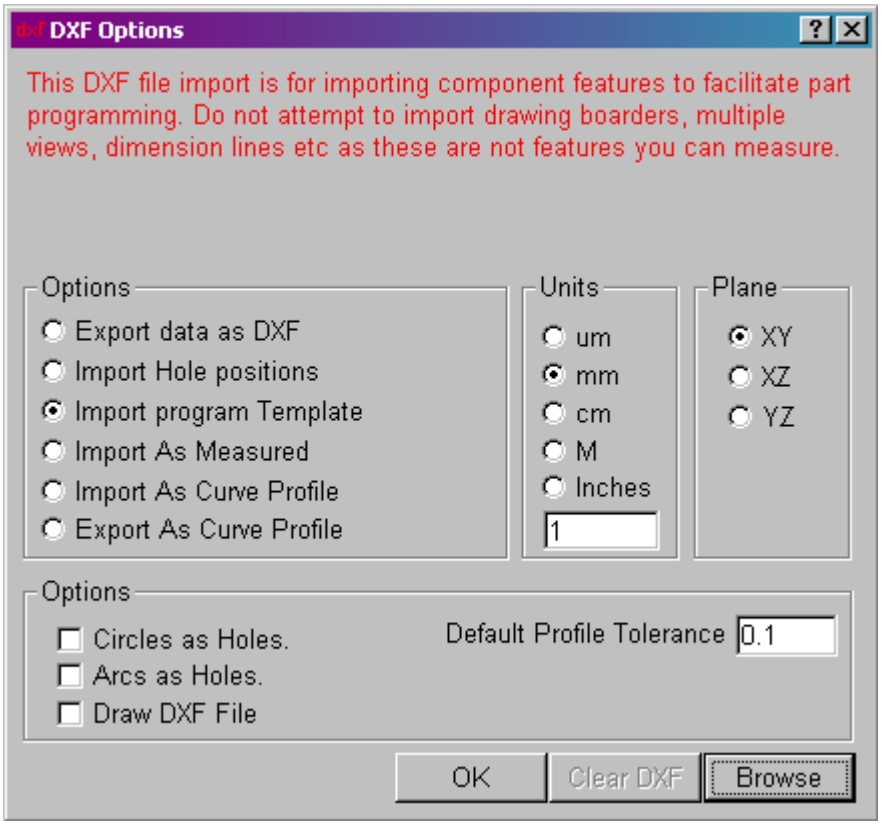
If the measurement window has not yet appeared, you will have to bring it up onto the screen by clicking on the relevant measurement button. Now erase the point as normal.

If however the erroneous point causes the wrong type of measurement window to appear on the screen, then you just have to click on the correct measurement icon and the points will be transferred to this type of feature measurement. However, please note that because this unit has been opened manually, it will have to be closed by clicking OK when finished.

27.0 DXF FILES

The Aberlink 3D software provides a full capability for the importing and exporting of DXF files. To bring up the DXF Options window click in the DXF button:

The following window will open:



27.1 Export Data as DXF

This option will allow the measured data to be exported as a DXF file. The geometry of the measured features can then be imported into a CAD package, for example for reverse engineering purposes.

Select the ‘Export Data as DXF’ option, and then use the ‘Browse’ button to save the DXF file where desired.

Note that this will export measured 2D features only. 3D shapes and the dimensions, which may have been called up are not compatible with a DXF file format.

27.2 Import Hole Positions

This function allows nominal hole positions to be imported from a DXF file.

Select the 'Import Hole Positions' option and browse to the relevant DXF File. The software will now match each measured hole to its corresponding position in the DXF file, so long as it is within the tolerance set under the 'DXF Hole Found Tolerance' setting under the 'Units' tab in the Machine Set Up window. The holes within this tolerance will temporarily change colour on the screen, and their nominal X,Y,Z centre positions will automatically be updated.

Note that the reference point (0,0,0 position) and alignment for the measurement must match that of the DXF file.

27.3 Import Program Template

This function can be used to import a DXF file, which can then be run as a program using the Play function. The features to be measured will appear as light grey on the Main Screen. When the program is run the feature to be measured will turn green, and its measurement window will automatically open.

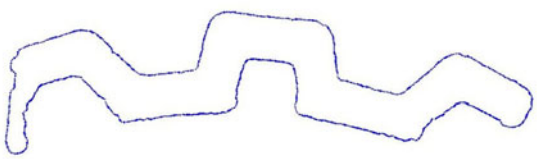
27.4 Import As Measured

This function exactly the same as above, except that with this option the features to be measured will appear as dark blue (measured features) rather than light grey (template).

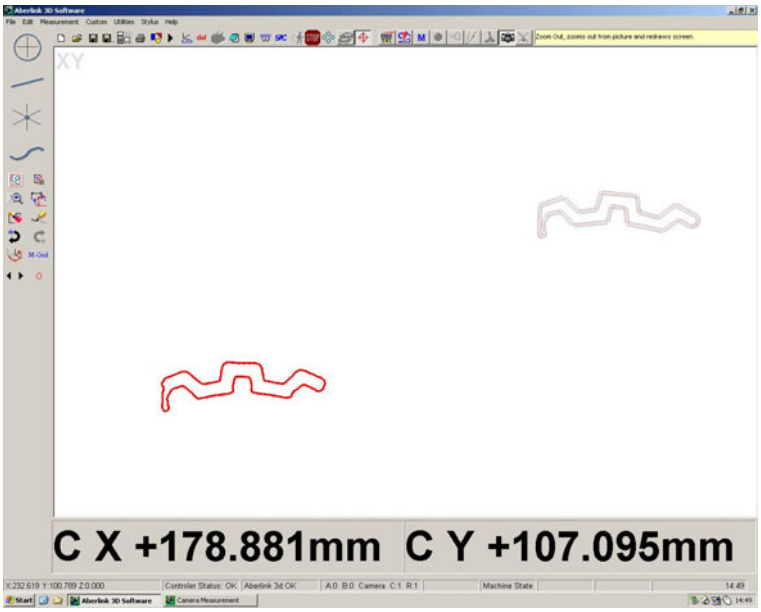
27.5 Import As Curve Profile

If the component shape has been measured as a Curve profile, it is possible to import the DXF file for the shape, and perform a best-fit optimisation to get the measured points to fit the DXF file.

Firstly measure the component as a Curve. This will appear as a series of blue dots in the Main Screen:

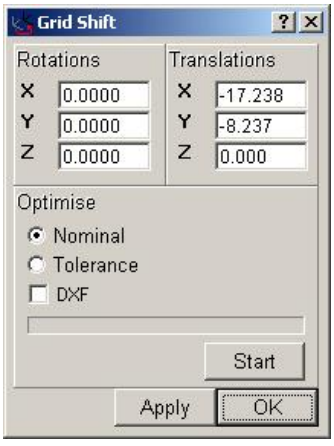


Select ‘Import As Curve Profile’ in the DXF window. Also enter the default profile tolerance required, and then browse to the DXF file. When you OK the DXF window the DXF file will be drawn in the Main Screen, but will not necessarily be in the same position as measured component, so it may be necessary to zoom out in order to see it:

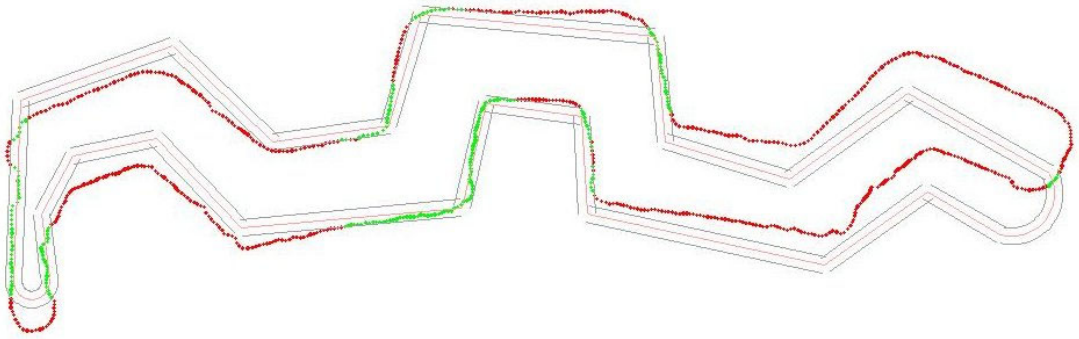


The nominal DXF file is shown as a red line, with the default tolerance shown as grey lines on either side.

Now open the Grid Shift window as follows:

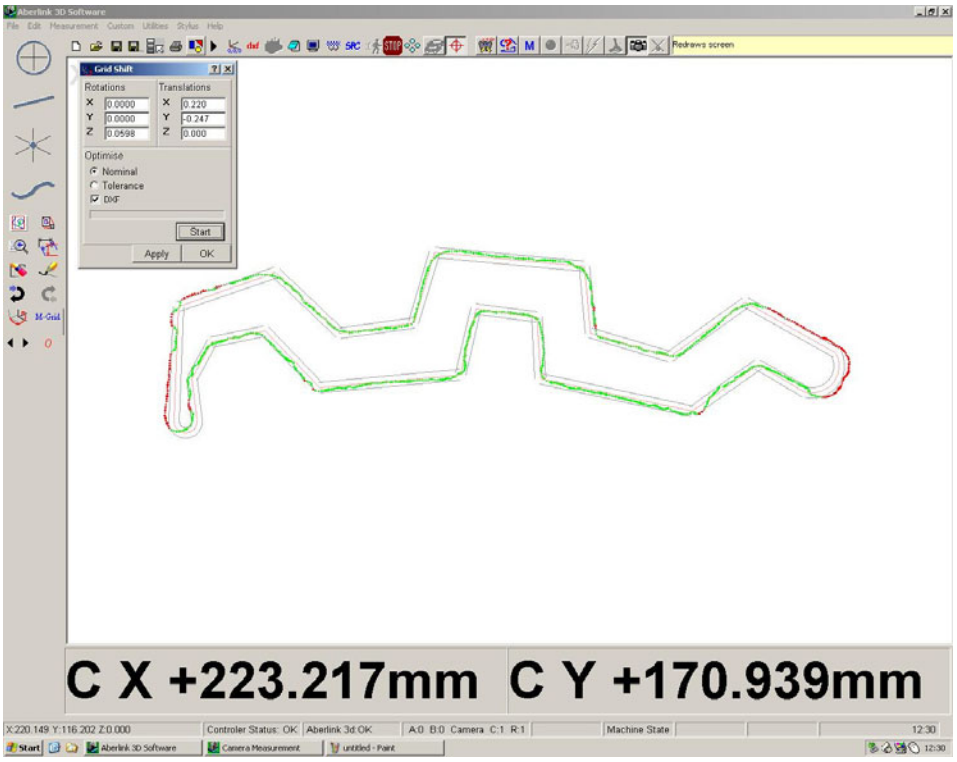


Whilst this window is open, the measured data points can be dragged using the right mouse button so that they are approximately over the top of the imported DXF file:



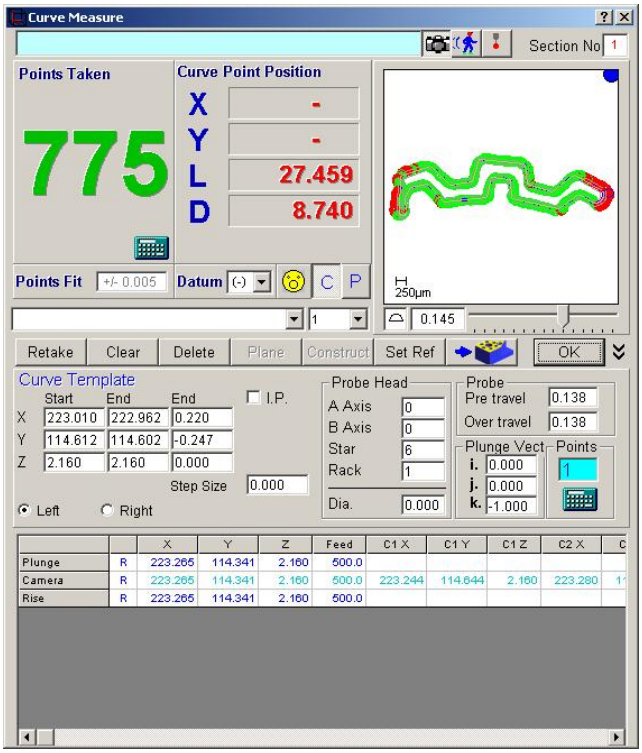
You will see that where the points are within the specified tolerance band they are shown in green, and where they are outside they are shown in red.

Now to perform a best fit, tick the DXF box under the ‘Optimise’ option and then click start. The software will now take a few seconds to calculate the rotation and translations required to fit the measured data to the DXF file. When it is finished the measured points will snap to the DXF file. Click OK and the software will return to the Main Screen:



At a glance the areas outside of the tolerance can be seen as they are shown in red.

Right clicking on the representation of the curve (or using the Feature Select buttons) will re-open the Curve Measure window:



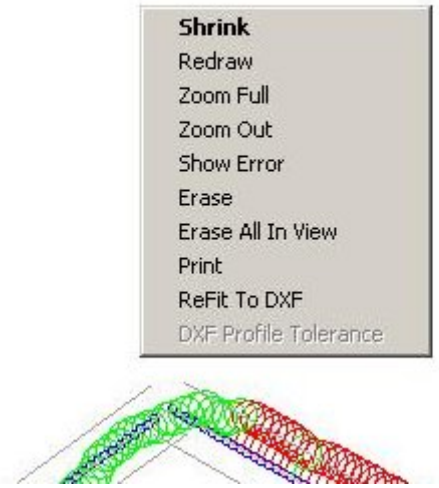
Right click in the graphic area and choose ‘expand’ to make the graphical representation fill the whole screen:

The measurement point shown in blue denotes the biggest error.

This value will be shown as the Profile of a Surface value in the bottom, left hand corner of the window.

The scale can be modified by using the slider bar, which is located next to the Profile of a Surface box. A scale indicator line is shown above the slider bar.

Right clicking in the window will bring up the following menu:



Shrink - will return to the full Curve Measure window.

Redraw - will refresh the image.

Zoom Full - will zoom to fit the whole of the Curve to fit the window.

Zoom Out - will zoom out by a fixed percentage.

Show Error - will bring up a box showing the error of any point right clicked on.

Erase - will erase any point right clicked on.

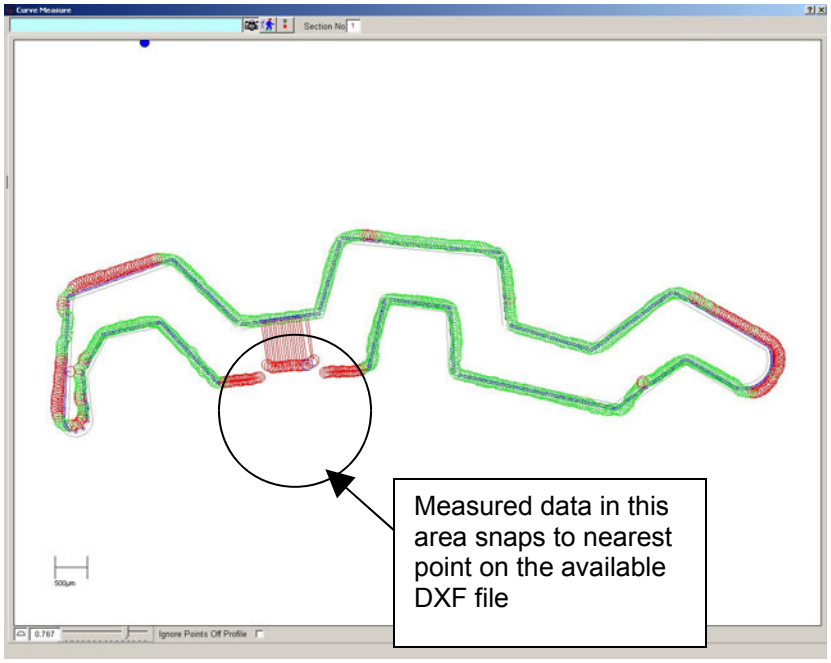
Erase All In View - will allow a box to be dragged over multiple points to be erased.

Print - will allow this representation to be printed as a Feature Profile report.

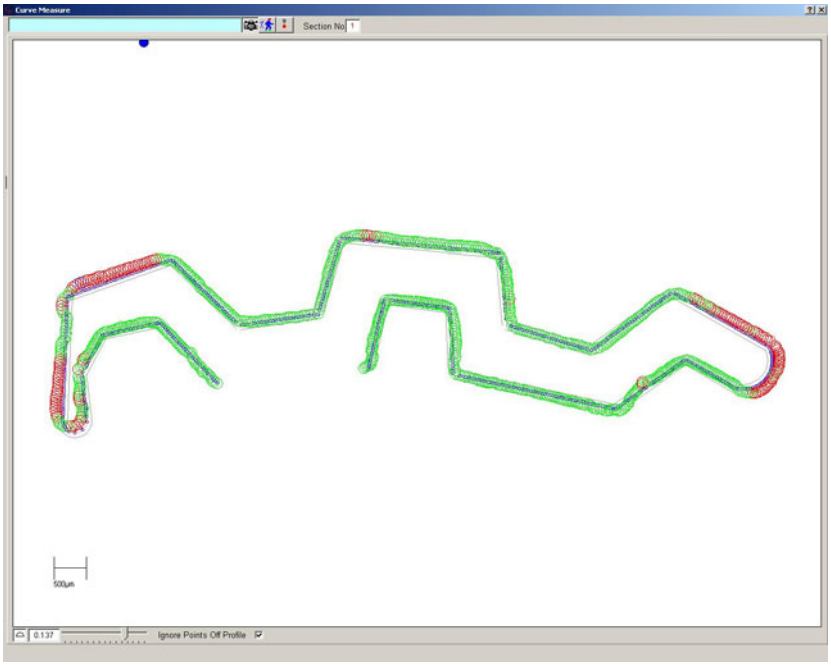
ReFit to DXF - will allow the best fit routine to be repeated.

DXF Profile Tolerance - will allow the default tolerance that was imported with the DXF file to be changed for any feature within the DXF file. Right click on that feature and select this option.

If the DXF file happens to contain gaps for any reason, this can produce errors because the error for the measured data within the gap(s) will be reported to the closest point on the DXF file, as follows:



In this situation simply tick the ‘Ignore Points Off Profile’ box, which will now produce a representation as follows:



27.6 Export As Curve Profile

When a DXF file is imported as a Curve Profile, the default tolerance will be applied to all features within the DXF file. It is possible to change the tolerance for each individual feature of the DXF file by right clicking on it and selecting the 'DXF Profile Tolerance' option, as described above.

If you wish to save the DXF file again complete with the modified tolerances, then the Export As Curve Profile option can be used.

28.0 DISPLAY MEASURED DETAILS OR TEMPLATE DETAILS

or

When a program is first made the software will create a template for each feature, that will be used for each subsequent measurement. Usually this information will be taken from each feature on the first time that is was measured, however, it is also possible to download a program template from a DXF file.

When the button shows ‘M’ the Main Screen will show measured features drawn in blue.

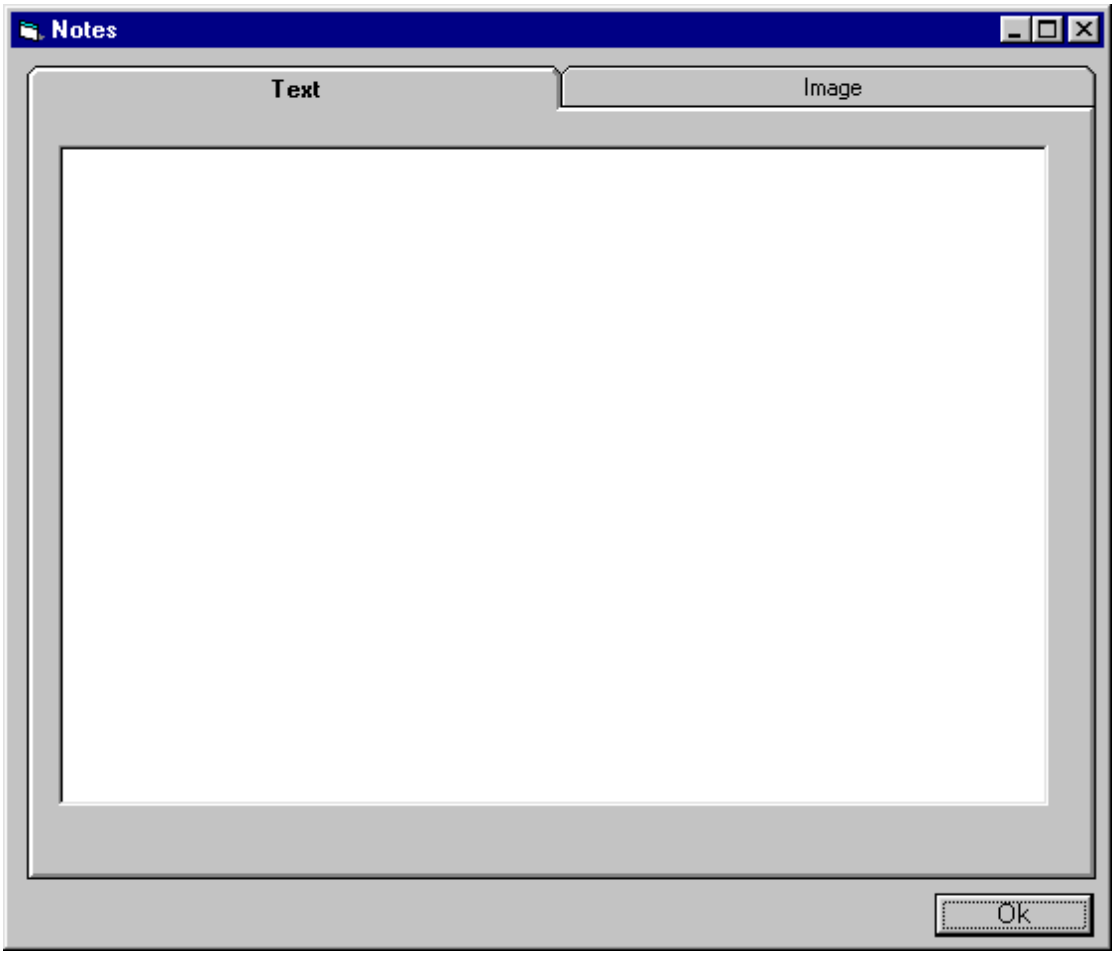
If you click on the ‘M’ the button will change to ‘T’ and the template for the inspection will now be shown in grey.

Click on the ‘T’ again to change it back to ‘M’.

29.0 INSPECTION NOTES

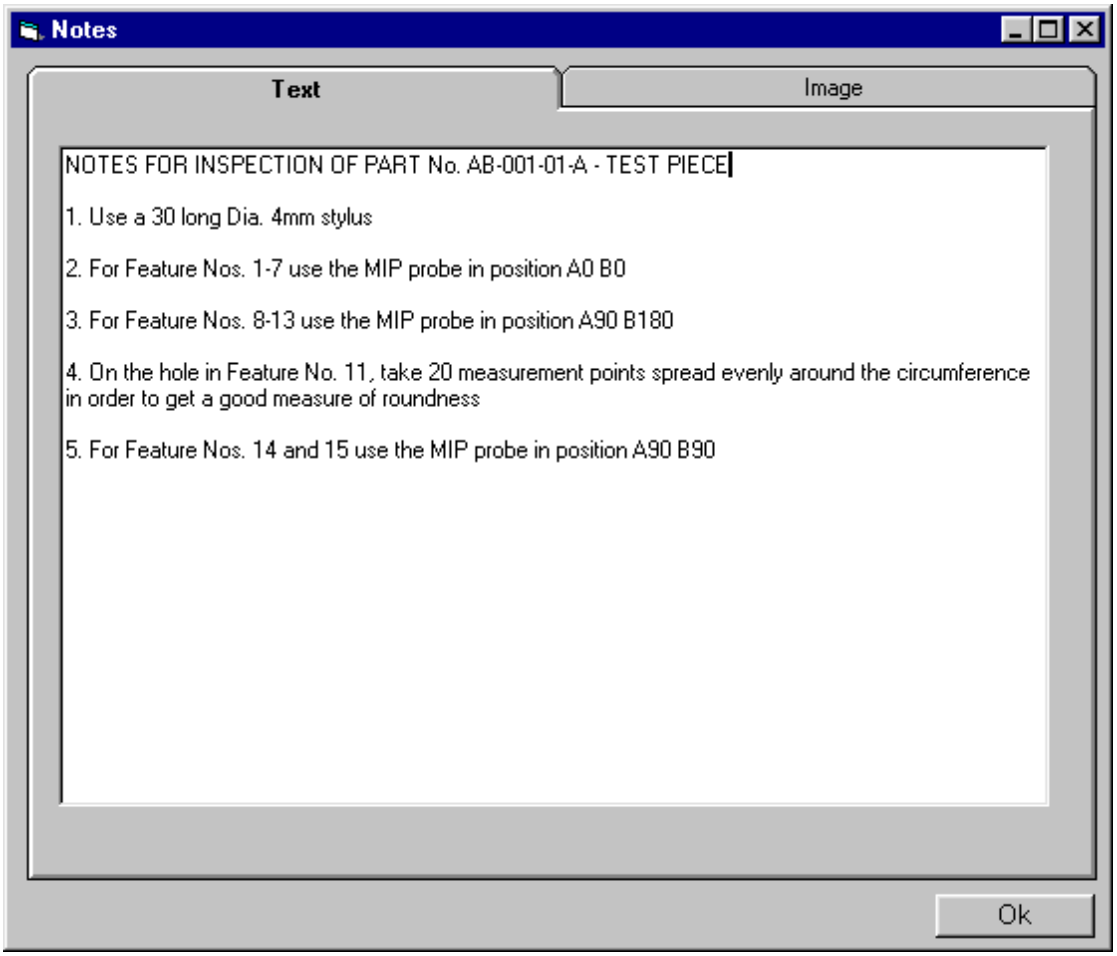
If you wish to save text or images with the inspection results, then click on the Inspection Notes button:

This will bring up the Inspection Notes window:



The inspection window shows two tabs ‘Text’ and ‘Image’. The text area allows any inspection notes to be added e.g. comments on the number of points to take on a particular feature, or the type of stylus to use.

To add text click on the ‘Text’ tab. And type in the required notes:


Notes
Text
Image
NOTES FOR INSPECTION OF PART No. AB-001-01-A - TEST PIECE
1. Use a 30 long Dia. 4mm stylus
2. For Feature Nos. 1-7 use the MIP probe in position A0 B0
3. For Feature Nos. 8-13 use the MIP probe in position A90 B180
4. On the hole in Feature No. 11, take 20 measurement points spread evenly around the circumference in order to get a good measure of roundness
5. For Feature Nos. 14 and 15 use the MIP probe in position A90 B90
Ok

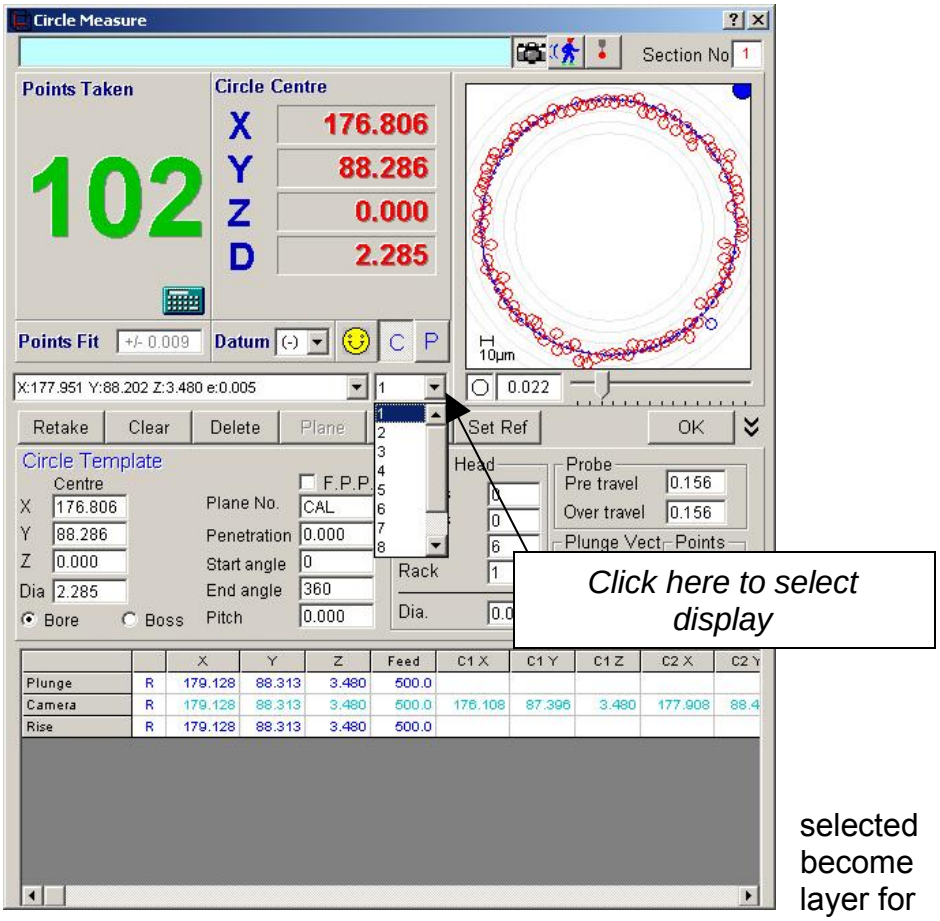
To import an image click on the ‘Image’ tab and then the ‘Load Image’ button. Browse to the picture file to upload it.

30.0 DISPLAY LAYERS

The main screen can occasionally become too cluttered to display information clearly. For this purpose the features and dimensions can be placed in up to 10 display layers, which may then be switched on and off to make the feature or dimension either visible or invisible, so that the information may be displayed more clearly.

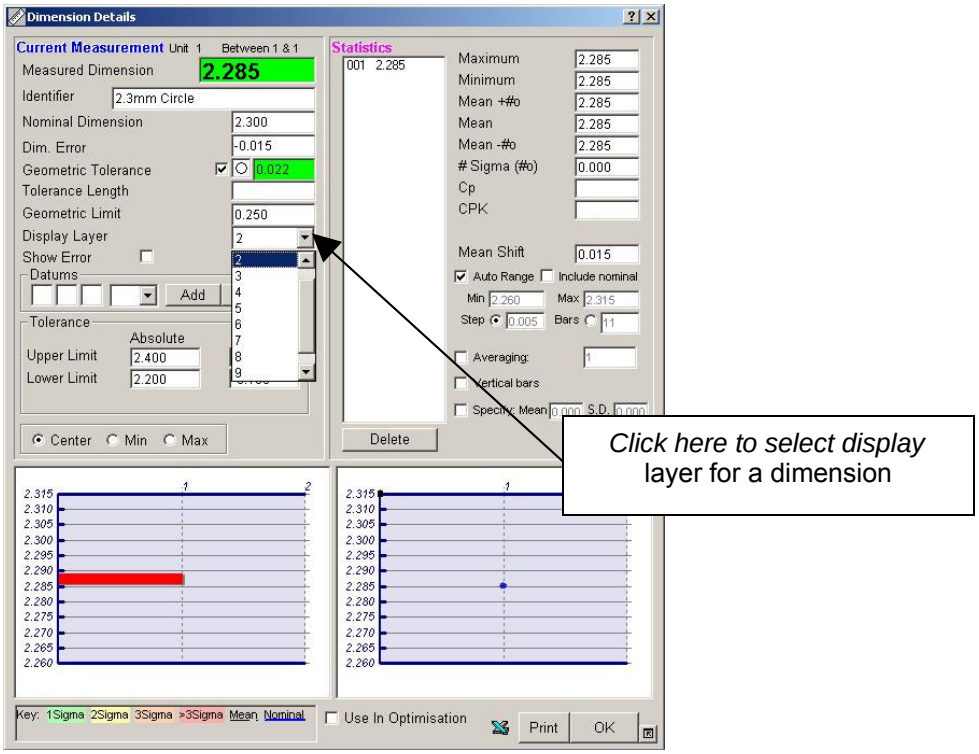
By default all features (Circle, Arcs, Points etc.) are placed into display layer 1 and all the dimensions are placed into layer 2.

To change the layer of a feature click on the Measurement Window as shown below:



The layer selected will become the default layer for any further features measured.

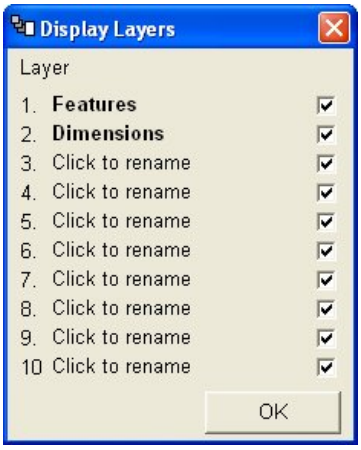
To change the layer of a dimension right click on the dimension to bring up the Dimension Details window:



Click on the black arrow adjacent to 'Display Layer' and select the desired layer. The layer selected will become the default layer for any further features measured.

To turn layers on and off click on the 'Display Drawing Layers' button:

This will bring up the following window:



Layers can be named by clicking on it and typing in the desired name.

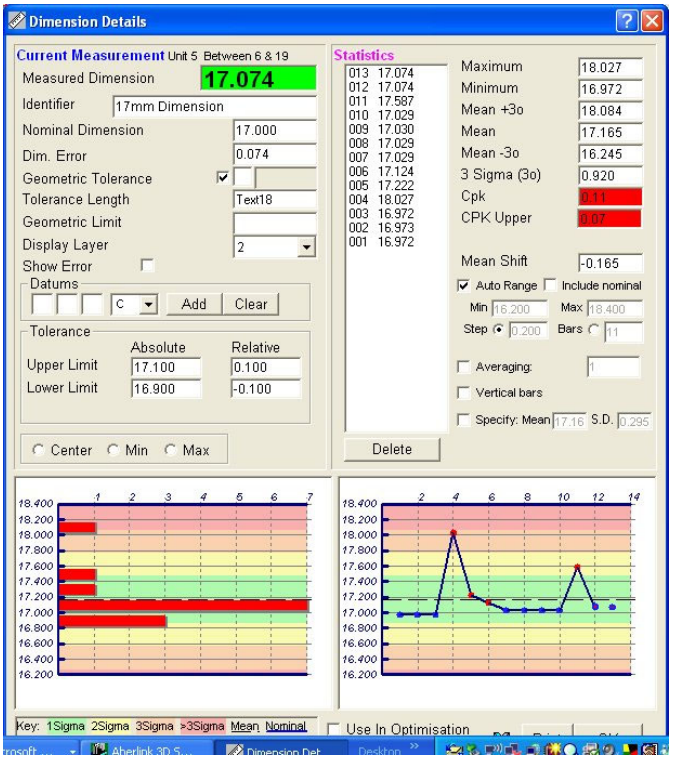
The default layers for both the features and dimensions are shown in bold type. You cannot switch these layers off. If you try to do this, the following warning will appear:

To change the default layer right click on the desired layer to bring up the following menu:

Having changed the default layer you will now be able to switch the original layer off.

31.0 SPC BATCH INFORMATION

If a batch of components have been measured using the Play function, and the SPC data has been collected using the 'SPC all Components' option, then the batch information for each dimension can be displayed by clicking on the small icon in the bottom left hand corner of the Dimension Details window:

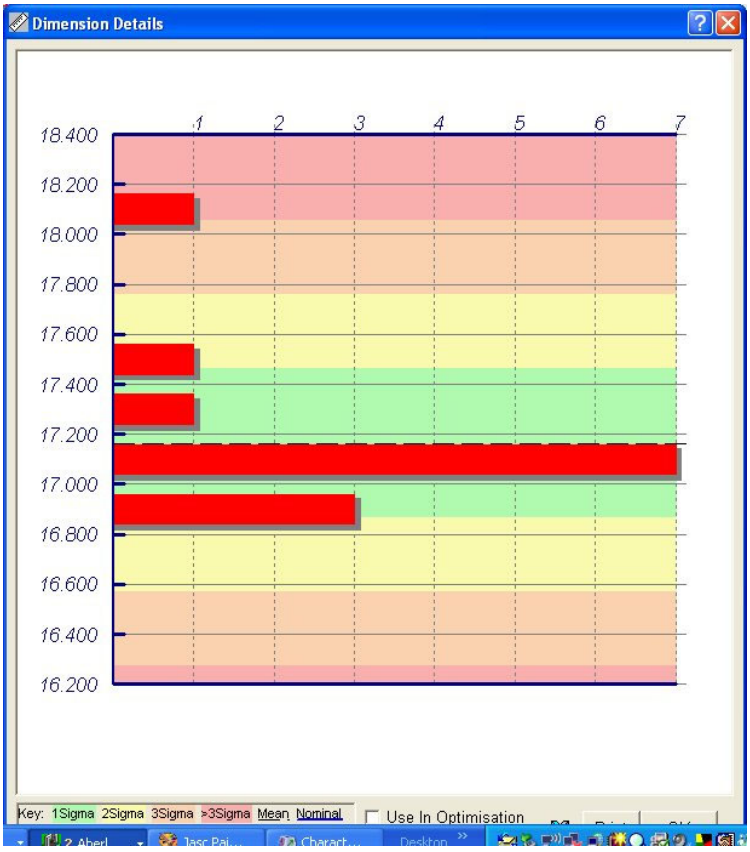


The measured value for all the components in the batch is listed in the white column.

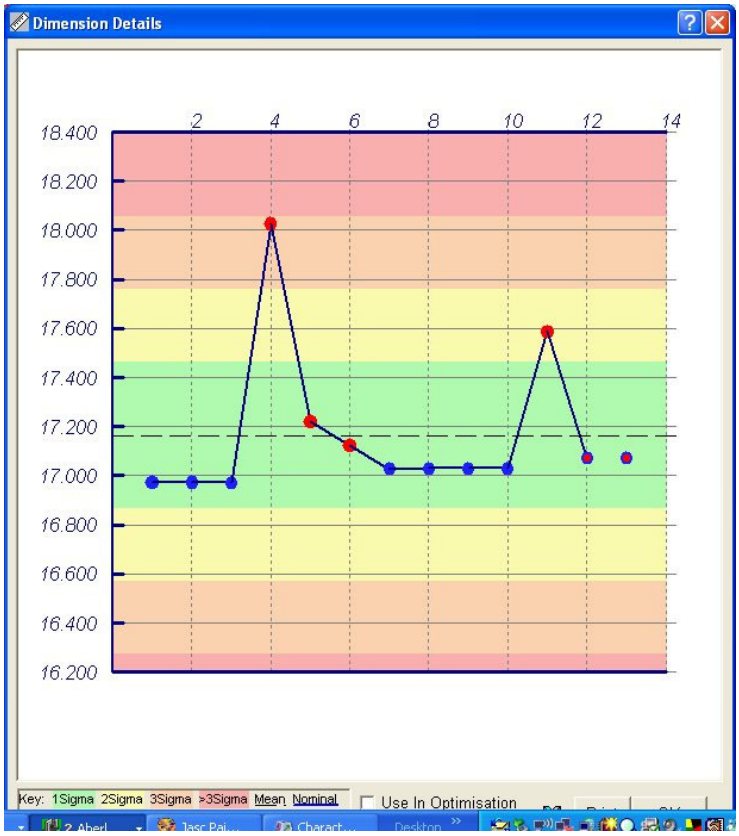
To the right of this column is listed the following batch information:

- Maximum
- Minimum
- Mean + 3σ (where the number of σ is configurable by the user)
- Mean (value for the batch)
- Mean - 3σ (where the number of σ is configurable by the user)
- 3 Sigma (3σ)
- CpK value
- CpK Upper
- Mean shift (difference between the mean and nominal values)

Below are shown two graphs of the measured data. Clicking either of these will expand it to fill the whole window:


Dimension Details
1 2 3 4 5 6 7
18.400
18.200
18.000
17.800
17.600
17.400
17.200
17.000
16.800
16.600
16.400
16.200
Key: 1Sigma 2Sigma 3Sigma >3Sigma Mean Nominal
Use In Optimisation

or


Dimension Details
2 4 6 8 10 12 14
18.400
18.200
18.000
17.800
17.600
17.400
17.200
17.000
16.800
16.600
16.400
16.200
Key: 1Sigma 2Sigma 3Sigma >3Sigma Mean Nominal
Use In Optimisation

Click on it again to return to the full Dimension Details window. The green band represents the 1σ band, yellow the 2σ, orange the 3σ and red greater than 3σ.

If you un-tick the ‘Auto Range’ box, the graphs may be customised as follows:

1) To include the nominal value.
2) Change the step size or number of bars in the bar graph.
3) Average through adjacent points (select number of points) in the scatter graph.
4) Change the bar graph to vertical bars with a normal distribution curve shown.
5) Specify the Mean and Standard deviation for the batch (rather than use the calculated values)

The ‘Delete’ button can be used to remove specified points from the batch. Select the measurement value from the list before clicking ‘Delete’

The ‘Print’ button will print a summary of the batch information.

If Excel has been loaded on to the Project X controller (not standard), then the batch information can be downloaded into an Excel spreadsheet by simply clicking on the Excel button within the window.

Clicking on the icon in the bottom right hand corner again will once again hide the SPC information.

32.0 EXPORTING SPC DATA

There is SPC batch information that can be stored and reported within the Aberlink 3D software as standard (see previous chapter).

However, it is also possible to output the measured data to certain external SPC programs, using the SPC button shown above.

To find out for which SPC packages an interface is available please contact Aberlink directly.